DESIGN BETTER PRODUCTS

SolidWorks® 2008

What's New

© 1995-2007, Dassault Systemes
SolidWorks Corporation is a Dassault Systemes S.A. (Nasdaq:DASTY) company.
300 Baker Avenue
Concord, Massachusetts 01742 USA
All Rights Reserved

U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,603,486; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,184,044; and foreign patents, (e.g. EP 1,116,190 and JP 3,517,643). U.S. and foreign patents pending.
The information and the software discussed in this document are subject to change without notice and are not commitments by SolidWorks.
No material may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose without the express written permission of SolidWorks.
The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license. All warranties given by SolidWorks as to the software and documentation are set forth in the SolidWorks Corporation License and Subscription Service Agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties.
SolidWorks, PDMWorks, 3D PartStream.NET, 3D ContentCentral, DWGeditor, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of SolidWorks.
SolidWorks 2008 is a product name of SolidWorks Corporation.
COSMOSXpress, DWGgateway, Feature Palette, PhotoWorks, TolAnalyst, and XchangeWorks are trademarks of SolidWorks.
COSMOS and COSMOSWorks are registered trademarks, and COSMOSMotion, COSMOSDesignStar, and COSMOSFloWorks are trademarks of Structural Research & Analysis Corp.
FeatureWorks is a registered trademark of Geometric Software Solutions Co. Ltd.
Other brand or product names are trademarks or registered trademarks of their respective holders.

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY
U.S. Government Restricted Rights. Use, duplication, or disclosure by the government is subject to restrictions as set forth in FAR 52.227-19 (Commercial Computer Software - Restricted Rights), DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), and in the license agreement, as applicable.
Contractor/Manufacturer:
SolidWorks Corporation, 300 Baker Avenue, Concord, Massachusetts 01742 USA
Portions of this software © 1990-2007 D-Cubed Ltd., © 1998-2007 Geometric Software Solutions Co. Ltd., © 1986-2007 mental images GmbH & Co. KG, © 1996-2007 Microsoft Corporation, © 1997-2007, Structural Research & Analysis Corp., © 2000-2007 Tech Soft 3D, and © 1998-2007 3Dconnexion, IntelliCAD Technology Consortium, Independent JPEG Group. All Rights Reserved.
Portions of this software are from PhysX™ by AGEIA, 2006-2007.
Portions of this software are copyrighted by and are the property of UGS Corp. © 2007.
Copyright 1984-2007 Adobe Systems Inc. and its licensors. All rights reserved.
Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending.
Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries. For more copyright information, see SolidWorks Help About.
Other portions of SolidWorks 2008 are licensed from SolidWorks licensors.
All Rights Reserved
PDMWorks Enterprise software is © 1998-2007 SolidWorks Nordic AB. All rights reserved.
Outside In® Viewer Technology © 1992-2007 Stellent Chicago, Inc.

Contents

Introduction

About this Book xii
Using This Book xiii
Converting Older SolidWorks Files to SolidWorks 2008 xiv

Chapter 1 User Interface

Menu Bar 1-2
Menu Bar Toolbar 1-2
Menu Bar Menus 1-2
SolidWorks Search and Help 1-2
CommandManager 1-3
Customizing the CommandManager 1-3
Activating SolidWorks Office Add-ins 1-3
FeatureManager Design Tree 1-3
Show/Hide FeatureManager Items 1-4
FeatureManager Design Tree Filter 1-4
Tags 1-6
Heads-up View Toolbar 1-6
Context Toolbars 1-6
Shortcut Bars 1-7
Opening and Displaying Documents 1-7
Browse Recent Documents 1-8
Browse Open Documents 1-8
Document Preview Tooltips 1-9
Task Pane 1-9
Flyout Tool Buttons 1-9
Control of Message Displays 1-10

Modifying Document Properties 1-10
Design Clipart 1-10

Chapter 2 RealView

RealView Workflow 2-2
Legacy Models 2-2
Appearances 2-3
Appearances PropertyManager 2-3
Scenes 2-8
Basic Scenes 2-8
Presentation Scenes 2-9
Studio Scenes 2-10
Edit Scene PropertyManager 2-11
Dynamic Highlighting 2-14

Chapter 3 Sketching

3D Sketch Symmetry 3-2
Blocks 3-3
Create 2D Blocks in 3D Sketches 3-3
Area/Hatch Fill 3-3
Consolidated PropertyManagers 3-4
Auto Trace Tools 3-5
Show/Hide Sketch 3-6
Sketching in Instant3D 3-6
SketchXpert 3-7
Enhancements 3-7
Splines 3-8
Continuity at Handles 3-8
Curvature Constraints 3-9
Spline Manipulators Available while Not Editing a Sketch 3-10
Spline on Surface 3-10

Chapter 4 Features

Boundary Surfaces 4-2
Linear Option 4-2
Tangent Influence 4-2
Fillets 4-3
Fillet Corners 4-3
Fillet Selection 4-3

Hole Series 4-4
Enhancements: 4-4
Hole Wizard 4-5
Enhancements 4-5
Instant3D 4-6
Patterns 4-8
Circular Patterns 4-8
Cosmetic Patterns 4-8
Split Lines and Parts 4-9
Split Lines 4-9
Split Parts 4-9
Sweeps 4-10

Chapter 5 Parts

Mate to Coordinate Systems 5-2
Inserting and Mirroring Parts 5-2
Inserting Sketches when Inserting Parts 5-2
Breaking the Link to a Part 5-2
Custom Properties of Parts 5-2
Automatic Positioning of Parts Using Mate References 5-3
Isolating Bodies in Part Mode 5-4

Chapter 6 Assemblies

General 6-2
Assembly Statistics 6-2
References 6-2
Stack-Up Analysis 6-2
Display States in eDrawings® 6-2
AssemblyXpert 6-2
Derived Components 6-3
Custom Properties of Mirrored Components 6-3
Derived Component Patterns 6-3
Hole Alignment 6-6
Layout-based Assembly Design 6-7
Virtual Components 6-7
Layout Sketches 6-8
Mates 6-9
Mate Icons in the FeatureManager Design Tree 6-9
Mating to Origins and Coordinate Systems 6-9

Mate PropertyManager 6-10
Copy with Mates 6-12
Selection 6-14
Selecting Sub-Assemblies in the Graphics Area 6-14
Selection Tools 6-14
Advanced Component Selection 6-14
Filtering the FeatureManager Design Tree 6-14
Show Hidden Components 6-16
Simplified Representations for Assemblies 6-17
Overview 6-17
Display States 6-18
Selective Loading of Components 6-18
Legacy Behavior 6-20
Smart Fasteners 6-21

Chapter 7 Configurations

General 7-2
Design Table Feature 7-2
Hole Wizard Holes 7-2
Modify Dialog Box 7-2
Creating a PropertyManager to Configure Components 7-2
Creating and Modifying Configurations 7-4
Parts 7-4
Assemblies 7-6

Chapter 8 Motion Studies

Motion Studies Interface 8-2
Levels of Functionality 8-2
MotionManager Enhancements 8-3
Collapsible Pane 8-3
Filters 8-3
Keys Points 8-3
Playback Speed 8-3
Save Animation 8-3
Assembly Motion 8-4
Adding Motors to Animations 8-4
Physical Simulation 8-4
Contacts 8-4
Springs 8-4

COSMOSMotion 8-4
Analytical Properties of Mates 8-4
Dampers 8-4
Fixed and Floating Parts 8-5
Legacy Studies 8-5
Plots 8-5
Redundant Constraint Handling 8-5

Chapter 9 Drawings and Detailing

General 9-2
Balloons in Notes 9-2
Balloon Text 9-2
Dialog Box Removal 9-2
Dimension Alignment 9-2
Dimension Properties 9-2
Leaders 9-2
Drawing Sheets 9-3
New Drawings from Open Documents 9-3
Copying Sheets 9-3
Inserting Images 9-3
Drawing Views 9-3
Broken Views 9-3
Section Views 9-4
Annotation Views 9-4
Sketch Entity Alignment 9-4
Bills of Materials 9-5
Column Contents 9-5
Property Modifications 9-5
Virtual Components 9-5
Weldment Material Usage 9-5
Tables 9-6
Editing Cells 9-6
Editing Cells Externally 9-6
Editing Tables 9-6
Equations in Cells 9-6
Fit Text to a Cell or Note 9-7
Header and Balloon Status 9-7
Header Position 9-7
Hide and Show Rows and Columns in Tables 9-7
Making Font Changes 9-7
Modifying Columns 9-8

Chapter 10 Dimensioning and Tolerancing

Overview 10-2
DimXpert for Parts 10-3
Features 10-3
Using DimXpert 10-4
TolAnalyst 10-11

Chapter 11 COSMOSWorks

General 11-2
Analysis Advisor 11-2
Hyperelastic Mooney-Rivlin and Ogden Material (A) 11-2
Shells Displayed by Thickness or Material 11-3
New Study Types 11-3
Pressure Vessel Study (P) 11-3
Linear Dynamic Study (A) 11-3
Nonlinear Dynamic Study (A) 11-8
Example of Linear Dynamic Study (A) 11-8
Analysis Studies 11-11
Beams 11-12
Design Scenarios 11-13
Large Displacement Support 11-13
Trend Tracker (P) 11-13
Loads and Restraints 11-14
Bolt Connectors 11-14
Pin Connectors 11-15
Mesh 11-15
Contact and Bonding 11-16
Stiffeners 11-17
Result Viewing 11-18
Design Insight Plots 11-19
Stress Linearization (P) 11-20

Chapter 12 Other Functionality

Installation 12-2
Application Programming Interfaces 12-2
DFMXpress 12-7
Rule Checks 12-7
Rule Configuration 12-8

DriveWorksXpress 12-8
Overview 12-9
Capturing Parameters 12-9
Creating Custom Properties 12-9
Designing Input Forms 12-9
Creating Rules 12-9
Running Models 12-10
eDrawings 12-11
Drawings 12-11
Mozilla Firefox 12-11
Previews 12-11
Pro/ENGINEER Files 12-11
RSS Feed 12-11
SolidWorks Display States 12-12
STL (stereolithography) Files 12-12
XPS (XML Paper Specification) Files 12-12
Import/Export 12-13
Adobe Illustrator 12-13
Adobe Photoshop 12-13
Autodesk Inventor 12-13
DXF/DWG Mapping 12-13
Sheet Metal Flat Patterns 12-13
Pro/ENGINEER 12-14
Rhino 12-14
XPS (XML Paper Specification) Files 12-14
SolidWorks Explorer 12-15
Right Pane Previews 12-15
Tags 12-15
Mold Design 12-16
MoldflowXpress 12-16
Sheet Metal 12-16
Adding Weld Beads to Sheet Metal Parts 12-16
Exporting Flat Patterns 12-16
SolidWorks Rx 12-17
Weldments 12-18
Transferring Cut List Information with Weldment Body 12-18
Bills of Materials 12-18
Welding Bodies With Gaps 12-18
Structural Member Profile Orientation 12-19

Chapter 13 SolidWorks Office Professional Components

FeatureWorks 13-2
Automatic Feature Recognition 13-2
Resize Tool 13-3
Base Sweeps with Internal Loops 13-4
PhotoWorks 13-5
Batch Document Processing 13-5
Cubic Environment Mapping 13-6
Indirect Illumination 13-7
Appearances and Scenes 13-9
Appearances PropertyManager 13-9
Emissive Appearances 13-10
PhotoWorks Scene Editor 13-11
System Options 13-13
Image Adjustment 13-14
Use Realistic Falloff 13-15
SolidWorks Design Checker 13-16
Check Documents from DWG Files 13-16
Criticality Level Checks 13-16
Font Checks Duplication 13-17
Multiple Standard Files on Active Document 13-17
Document Checks 13-18
Dimension Checks 13-18
Drawing Document Checks 13-19
Part Document Checks 13-19
Assembly Document Checks 13-20
Feature Checks 13-20
SolidWorks Task Scheduler 13-20
Export PDMWorks Files 13-20
Render and Animation 13-21
Dissect Files 13-21
SolidWorks Utilities 13-22
Compare Documents - Assemblies 13-22
Feature Paint 13-22
Find Replace Annotations 13-22
Simplify Enhancements 13-22
Symmetry Check 13-23
Thickness Analysis 13-23

PDMWorks Workgroup 13-24
Portable Document Format (PDF) Support 13-24
Viewing Changed Document Properties with eDrawings 13-25
PDMWorks Workgroup Vault Import and Export 13-25
Toolbox 13-26
Auto Size Toolbox Components 13-26
Content 13-27

Chapter 14 SolidWorks Office Premium Components

ScanTo3D 14-2
General 14-2
Deviation Analysis 14-5
Mesh Editing Tools 14-6
Curve Wizard 14-7
Mesh Prep Wizard 14-8
Surface Wizard 14-10
SolidWorks Routing 14-14
General 14-14
Electrical Connectors with Multiple Connection Points 14-14
Fixed Length 14-14
Lightweight Connection Points and Route Points 14-15
Stock Lengths for Pipes 14-15
Options 14-15
Auto Route 14-16
Flattened Route 14-16
TolAnalyst 14-17

Introduction

About this Book

This book highlights and helps you learn the new functionality in the SolidWorks® 2008 software. It introduces concepts and provides step-by-step examples for many of the new functions.

This book does not cover all details of the new functions in this software release. For complete coverage, refer to the *SolidWorks Help*.

Intended Audience

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

Additional Resources

Interactive What's New is another resource where you can learn about the new functionality of the SolidWorks software. Click next to new menu items and the title of new and changed PropertyManagers to read what is new about the command. A help topic appears with the text from this manual.

Late Changes

This book might not include all of the enhancements in the SolidWorks 2008 software. Late changes are documented in **SolidWorks Release Notes**.

Using This Book

Example Files

Use this book with the example part, assembly, and drawing files provided. The example files are placed in the <*install_dir*>**samples****whatsnew** folder. Because some of the example files are used with more than one example, they are installed as read-only to help you avoid overwriting them.

New in SolidWorks 2008: Some procedures are provided in separate tutorial-like files called *Hands-on Examples*. If a section of the this book has a related *Hands-on Example*, a hyperlink is provided so you can click to access it.

Conventions Used in this Book

Convention	Meaning
Bold	Any SolidWorks tool, menu item, or example file
Italic	Refers to books and other documents, or emphasizes text
	Tip
	References *SolidWorks Help*
Blue Underlined	Hyperlinks to *SolidWorks Help* or a *Hands-on Example*

Converting Older SolidWorks Files to SolidWorks 2008

Opening a SolidWorks document from an earlier release might take extra time. After the file is opened and saved, subsequent opening time returns to normal.

You can use the SolidWorks Conversion Wizard to automatically convert all of your SolidWorks files from an earlier version to the SolidWorks 2008 format. To access the Conversion Wizard, click Windows **Start**, then **All Programs**, **SolidWorks 2008**, **SolidWorks Tools**, **Conversion Wizard**.

Two report files are created in the conversion folder:

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

After you convert files to SolidWorks 2008, you cannot open them in older SolidWorks versions.

1

User Interface

This chapter describes enhancements to the user interface in the following areas:

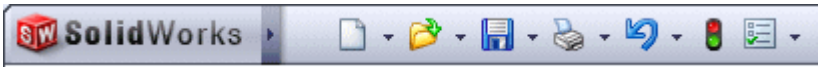
- Menu Bar
- CommandManager
- FeatureManager® Design Tree
- Tags
- Heads-up View Tools
- Context Toolbars
- Shortcut Toolbars
- Recent Document Browser
- Open Document Browser
- Task Pane
- Flyout Tool Buttons
- Control of Message Display
- Design Clipart

Menu Bar

The SolidWorks 2008 user interface has been redesigned to make maximum use of space. In addition to the title of the current document, a new Menu Bar contains a subset of tools from the Standard toolbar, the SolidWorks menus, the SolidWorks Search oval, and a flyout menu of Help options.

Menu Bar Toolbar

In the default view of the Menu bar, only the toolbar buttons are visible.

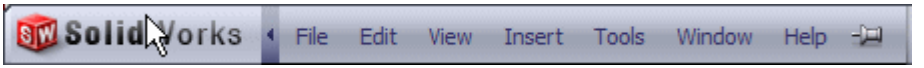


You can customize this toolbar in the same way that you customize toolbars in earlier versions of SolidWorks.

See **Customize Commands** in the help.

Menu Bar Menus

By default, menus are hidden. To display them, move the mouse over or click the SolidWorks logo.



To keep the menus visible, pin the menus open.

All menu items are now shown by default. You can customize menus to hide options you do not use.

See **Customize Menus** in the help.

SolidWorks Search and Help

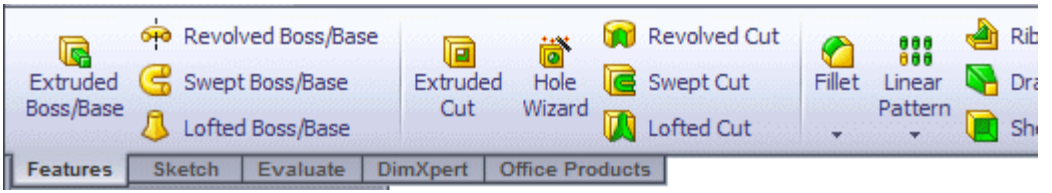
The SolidWorks Search oval is now located on the right side of the Menu Bar, along with a flyout menu of Help options.

The **SolidWorks Search** tool includes graphics (when available) of the items found during the search. Search results are shown in the Search view of the Task Pane. As with **Toolbox** items, the search items can be dragged into the graphics area to add to your model. The search results can include items from 3D ContentCentral® supplier catalogs.

CommandManager

The CommandManager, when displayed, is always docked above the graphics area.

Tabs below the left side of the CommandManager let you change the display of commands: these replace the control area buttons from previous SolidWorks versions. The tabs that are displayed by default depend on the type of document open and the work flow customization you have selected.



Customizing the CommandManager

You can customize CommandManager tabs by:

- Adding custom tabs and tool buttons
- Changing tool button labels
- Displaying or hiding tabs

See **CommandManager** in the help.

Activating SolidWorks Office Add-ins

If you have SolidWorks Office, SolidWorks Office Professional, or SolidWorks Office Premium, the **Office Products** tab appears on the CommandManager.

Use the **SolidWorks Office** flyout to activate SolidWorks Office add-ins installed on your computer and display their most frequently used commands.

FeatureManager Design Tree

New commands let you control what is displayed in the FeatureManager design tree.

You can:

- Show or Hide FeatureManager items
- Filter the FeatureManager design tree

In addition, new FeatureManager design tree icons distinguish between variants of features. For example, mates now have separate icons to denote what type they are. See **Mate Icons in the FeatureManager Design Tree** on page 6-9.

Show/Hide FeatureManager Items

You can control the visibility of items such as Design Binder and Equations .

To set the visibility of items in the FeatureManager tree:

1. Click **Options** (Menu bar), or **Tools, Options**.
2. On the **System Options** tab, click **FeatureManager**.
3. Under **Hide/Show Tree Items**, for each item, select one of the following:
   - **Automatic**. Displays the item if present. Otherwise, it is hidden.
   - **Hide/Show**. Always hides or shows the item.
4. Click **OK**.

In the FeatureManager design tree, you can access hidden items by right-clicking the top-level tree icon and selecting **Hidden Tree Items**.

You can also access the hide/show options by expanding the shortcut menu and selecting **Hide/Show Tree Items**.

See **FeatureManager Options** in the help.

FeatureManager Design Tree Filter

The FeatureManager design tree filter lets you search for specific features of parts and components of assemblies.

You can filter by:

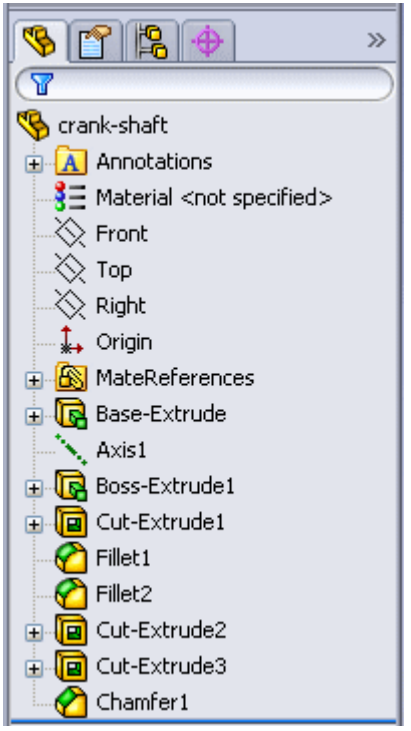
- Types of features
- Feature names
- Sketches
- Folders
- Mates
- User-defined tags
- Custom properties

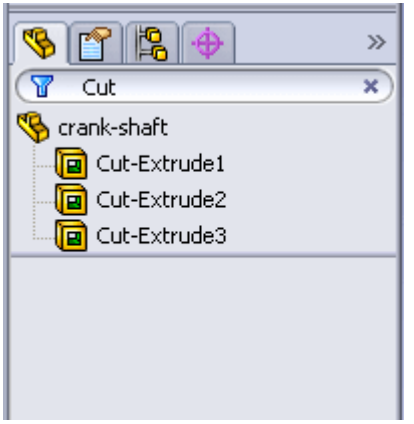
To filter the FeatureManager design tree:

1 In the FeatureManager, in the filter field, type a keyword.

In an assembly, click the down arrow to select additional parameters, such as setting the graphic area to display only the items matching the filter criteria. See **Filtering the FeatureManager Design Tree** on page 6-14.

2 To redisplay all features, click in the filter field.





See **Filtering the FeatureManager Design Tree** in the help.

Tags

Tags are keywords you add to SolidWorks documents and features to make them easier to filter and search.

- To aid in filtering the FeatureManager tree, add tags to selected features in the graphics area.
- To facilitate searching, add tags to selected documents:
  - On the File Explorer tab in the task pane
  - In the File Explorer pane in SolidWorks Explorer

See **Tags** in the help.

Heads-up View Toolbar

A transparent toolbar in each viewport provides all the common tools necessary for manipulating the view.

You cannot hide or customize the Heads-up View toolbar.

Custom and camera views you define appear on the **View Orientations** flyout .

The **View Filters** flyout lets you control the visibility of multiple graphics area items such as annotations and sketch relations, at the same time.

See **Heads-up View Tools** in the help.

Context Toolbars

When you select items in the graphics area or FeatureManager design tree geometry, context toolbars appear and provide access to frequently performed actions for that context, for example, editing the sketch of a selected face.The tools in the context toolbar are a subset of items previously found on the shortcut menus. You can still right-click while the context toolbar is displayed and see the additional menu items that relate to the currently selected item.

Context toolbars are available for the most commonly used selections. For the use of a context toolbar in drawing tables, see **Editing Tables** on page 9-6.

See **Context Toolbars** in the help.

Shortcut Bars

Customizable shortcut bars let you create your own set of “non-context” commands for each of the following modes:

- Part
- Assembly
- Drawing
- Sketch

You display these bars by pressing a user-definable keyboard shortcut. The default shortcut is the “S” key.

See **Shortcut Bars** in the help.

To customize the shortcut bar:

1. With nothing selected in the graphics area, press **S**.

2. Right-click the default shortcut bar that appears, and select **Customize**.
3. To add commands, on the **Commands** tab, select categories and drag tools to the shortcut toolbar.
4. With the **Customize** dialog box open, you can also:
   - Remove a tool by dragging it from the shortcut bar.
   - Resize the shortcut bar by moving the pointer over an edge and dragging it.
   - Change the keyboard shortcut by clicking the **Keyboard** tab, sorting by **Command**, scrolling to **Shortcut Bar**, and changing the value for **Shortcut(s)**.
5. Click **OK** to close the **Customize** dialog box.

Opening and Displaying Documents

New features make it easier to select the document to open or display.

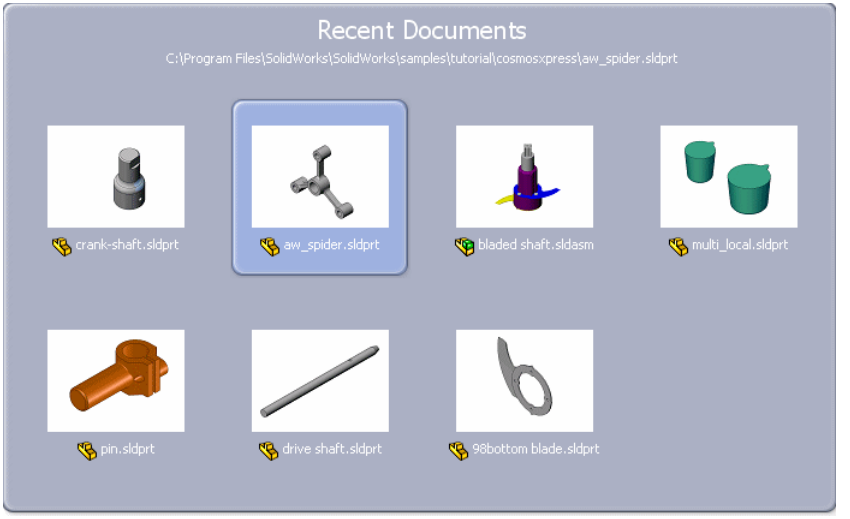
See **Opening and Displaying Documents** in the help.

Browse Recent Documents

You can select the document to activate or load either by name or visual preview in the Recent Document browser.

To visually select a document from recently viewed documents:

1 Click **File, Browse Recent Documents** or press **R** on the keyboard.



2 In the browser, move the pointer over the preview to display the full path to the document.

3 To open the document, click the preview.

To close a browser without selecting a document, click outside it or press **Esc**.

Browse Open Documents

You can browse currently open documents and select one to display.

To select a document:

1 Click **Window**, **Browse Open Documents**, or press and hold **Ctrl** and press **Tab**.

2 To scroll through the documents in the browser, move the pointer over the previews or press **Tab** (continue to hold **Ctrl**).

The preview is highlighted and the document's full path appears at the top of the browser.

3 To select the document, click the preview or release **Ctrl**.

Document Preview Tooltips

When you mouse over a document name in the File menu recent document list or the Windows menu open documents list, a preview tooltip appears.

To open or display the document, click the document name.

Task Pane

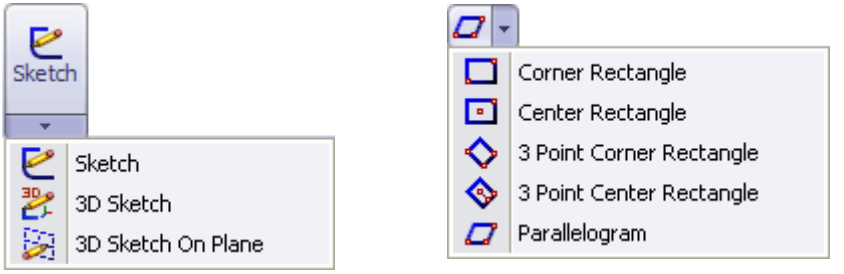
The behavior of the Task Pane has changed to provide additional space in the graphics area.

See **Task Pane** in the help.

Although the Task Pane can be undocked and moved around the screen, it can only be docked on the right of the SolidWorks window, with the Task Pane tabs extending to the left when it is docked and above when it is undocked.

Flyout Tool Buttons

Similar commands are grouped into flyout buttons on toolbars and the CommandManager. For example, variations of the rectangle are grouped together in a button with a flyout control.



When you click the flyout button without expanding:

- For some commands such as **Sketch**, the most commonly used command is performed. This command is the first listed and the command shown on the button.
- For commands used to sketch shapes such as rectangles, circles, and ellipses, it is now easier to repeatedly create the same shape variant. When you create a shape, the button icon changes to that shape. If you click the button again without expanding the flyout, the most recently used command is performed.

  For example, the default rectangle icon and command is a corner rectangle. If you sketch a parallelogram, the button icon changes to a parallelogram. The next time you sketch a rectangle, the default shape is a parallelogram.

See **Flyout Tool Buttons** in the help.

Control of Message Displays

You can now suppress many messages by checking **Don't ask me again** when a message appears.

If you later decide that the message should be displayed, you can reactivate it.

See **Advanced System Options** in the help.

To reactivate a suppressed message:

1. Click **Options** (Menu bar toolbar), or **Tools, Options**.
2. On the **System Options** tab, click **Advanced**.
3. Under **Dismissed messages (checked messages will be shown again)**, select the message that you want to display.

Modifying Document Properties

You can modify SolidWorks document properties without opening the document or even SolidWorks itself. Windows Explorer, SolidWorks File Explorer, or SolidWorks Explorer can be used to modify a document's **Custom** and **Summary** properties.

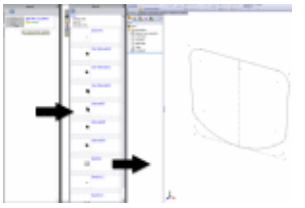
With SolidWorks File Explorer or Windows Explorer:

1. With the document closed, right click the file name and select **Properties**.
2. You can edit the properties on the **Custom** and **Summary** tabs. Your edits are present the next time the document is opened. See **Property Modifications** on page 9-5.

With SolidWorks Explorer PDMWorks Workgroup add-in:

1. Select a document in the SolidWorks Explorer vault pane.
2. In the right pane, on the **Properties** tab, double-click the value of an existing property and modify it.

Design Clipart



Design Clipart allows you to reuse sketches, features, views, and tables from SolidWorks models and drawings and data from DWG and DXF files. Design Clipart dissects SolidWorks, DWG, and DXF files and extracts data so it is reusable in SolidWorks.

See **Instant3D** on page 4-6 and **Dissect Files** on page 13-21.

2

RealView

This chapter describes enhancements to graphics in the following areas:

- RealView Workflow
- Appearances
- Scenes
- Dynamic Highlighting

RealView Workflow

On **RealView**-compatible systems, you can add **Appearances** and **Scenes** to display photo-realistic models and environments.

- **Appearances**. Materials are called **Appearances**. The appearance of a model is different from its physical properties. For example, you can assign a model the physical property of stainless steel, but apply the appearance of glossy car paint.
- **Scenes**. Scenes affect the way appearances look by displaying different environments that include reflective floors, photos that encompass the model, and reflections of backgrounds.

Verify that you have the latest drivers installed. Some **RealView**-compatible graphic cards may not display all of the effects (self-shadows and reflections).
See: www.solidworks.com/pages/services/videocardtesting.html.

With **RealView** off, you can still apply textures and colors using **Materials** and **Colors**. However, the preferred workflow with **RealView** on, is to use **Appearances** and **Scenes** from the Task Pane.

Legacy Models

SolidWorks 2007	RealView On	RealView Off
SolidWorks colors and textures	**Appearances** uses default plastic	SolidWorks colors and textures
SolidWorks colors and textures and PhotoWorks™ appearances	PhotoWorks appearances override SolidWorks colors and textures	SolidWorks colors and textures

To use RealView:

From the Heads-up View toolbar:

- Expand **View Settings** and click **RealView Graphics** to toggle **RealView**.
- Click **Apply scene** to cycle through scenes (see **Scenes** on page 2-8) and apply the next consecutive scene. The scenes are organized in the same order as in the **Scenes** folder on the Task Pane.
- Expand **Apply scene** and select a scene to apply it.

From the Task Pane, select the **RealView** tab to display:

- **Appearances**.
- **Scenes**.

You can also add backgrounds through **Options** , **Colors**.

Appearances

To apply appearances:

1. Expand the **Appearances** tab under **RealView** in the Task Pane.
2. Browse through the appearance folders on the top pane.
3. Select a preview from the bottom panel and:

Drag an appearance to:	To apply the appearance to:
the graphics area.	the entire part.
pre-selected features or body entities in the FeatureManager design tree.	the features or bodies.
pre-selected model faces.	the selected model faces.

4. To modify default appearances, press **Alt** to display the **Appearances** PropertyManager with the **Color/Image** and **Mapping** tabs.

> To apply appearances without displaying the **Appearances** PropertyManager, follow steps 1 through 3.

Appearances PropertyManager

Color/Image Tab

The **Color/Image** tab controls selection and color properties.

- **Selected Geometry**. Selects geometry with filters and removes appearance.
- **Color**. Changes the color.

Mapping Tab

The **Mapping** tab controls the orientation of the appearance (for example the direction of wood grain). Appearances without texture or pattern, such as **Gloss Glass**, have no mapping.

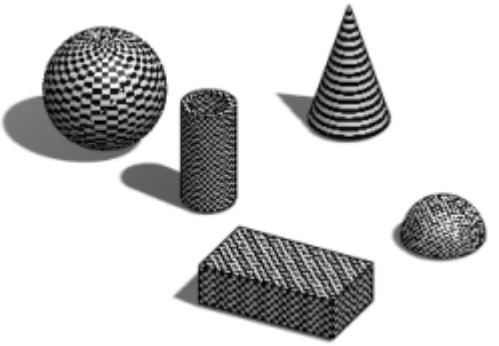
Mapping Style

Box mapping		Multiple-sided models
Surface mapping		Faces
Planar mapping		Planar faces
Spherical mapping		Spherical models
Cylindrical mapping		Cylindrical models

- **Axis direction**. Adjusts the projection direction of appearances, and maps them based on coordinates (**xy**, **zx**, or **yz**), the **Current View** (**Isometric**, **Front**, etc.), or a model's **Selected Reference** (faces, edges, etc.).
- **Rotation**. Adjusts the angle of the mapping.

The default mapping style is based on model geometry. The checker board appearance was applied to all models shown below. Only the mapping style was changed. You usually need to adjust the mapping style and mapping size (see **Mapping Size** on page 2-6) to optimize the model's appearance,

Box mapping



Surface Mapping

Planar Mapping

Spherical Mapping

Cylindrical Mapping

Mapping Size

Small mapping size

Regular mapping size

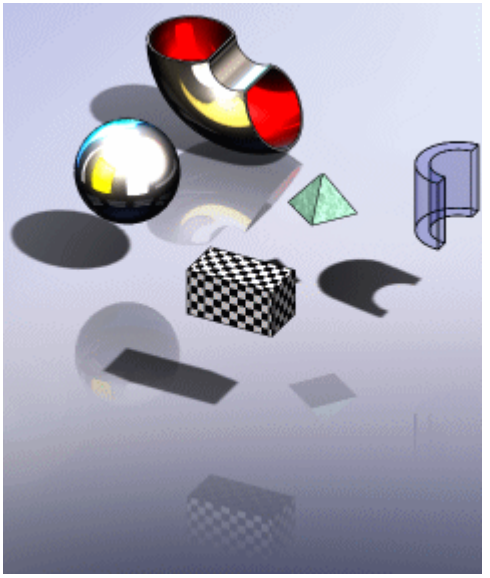
Big mapping size

To edit appearances:

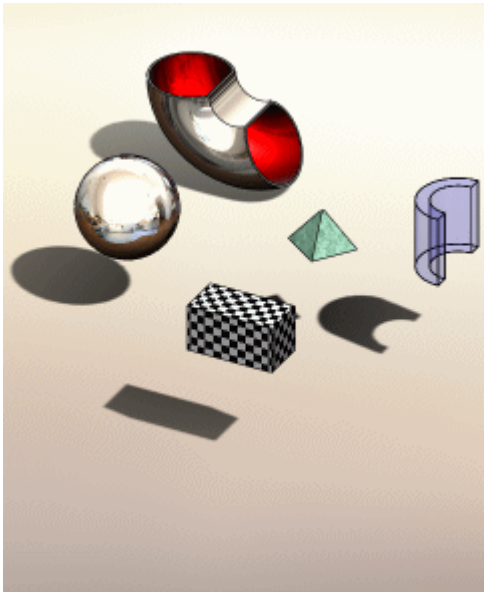
1. Right-click a model in the graphics area to display the context menu.

2. On the context toolbar, expand the **Appearance Callout**
3. In the **Appearances Callout**, click either block ☒ adjacent to the entity you want to modify.
4. In the **Appearances** PropertyManager, select the **Color/Image** or **Mapping** tab, and apply the changes.
5. Click ✔.

To display realistic reflections on shiny appearances such as chrome or high gloss paints, you must add a scene that includes an environment.



Default **RealView** background with reflective floor and no scene applied. The reflections are generic.



Warm Kitchen scene applied (non reflective floor). The reflections represent a real scene.

See **Appearances** in the help.

Scenes

Basic Scenes

This scene type is characterized by a simple background and lighting. Examples include **Warm Kitchen** and **Office Space**.

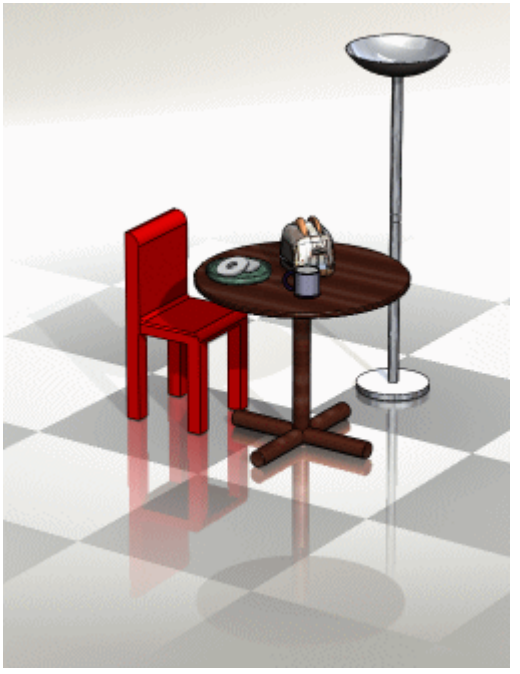
To modify the position of shadows, expand **Lights**, **Cameras and Scene** , and change the position of the first **Directional** light in the folder.

Presentation Scenes

This scene type is characterized by a background that becomes part of the environment. Examples are **Wood Floor Room** and **Courtyard Background**.

Studio Scenes

Studio Scenes combine elements from the previous two types. Examples are **Reflective Floor Checkered** and **Grill Lighting**.



To apply scenes:

- Expand **Scenes** in the **RealView** Task Pane, then drag a preview into the graphics area.
- Click **Apply Scene** (**View** toolbar) to apply the next consecutive scene. For example:

  If the current scene is **Warm Kitchen** and you click , you apply **White Kitchen** .
- Expand to display all scenes. Select a scene, and then click to apply that scene.

Hands-on Example

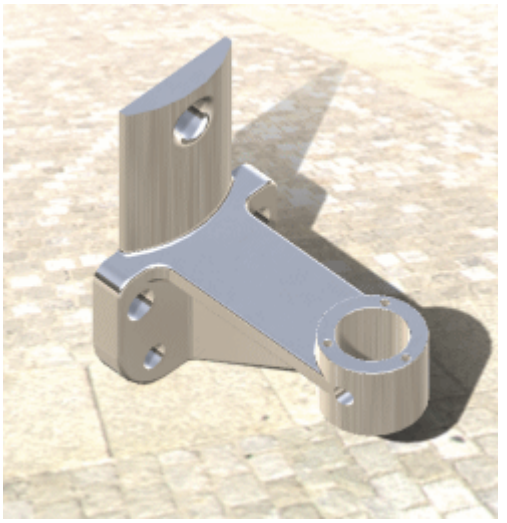
Edit Scene PropertyManager

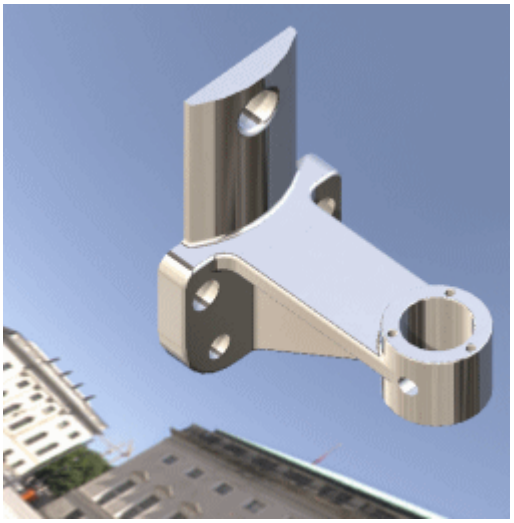
To edit scenes:

1. In the FeatureManager design tree, expand **Lights, Cameras and Scene** .
2. Double-click **Scene** to open the **Edit Scene** PropertyManager.

Floor Location

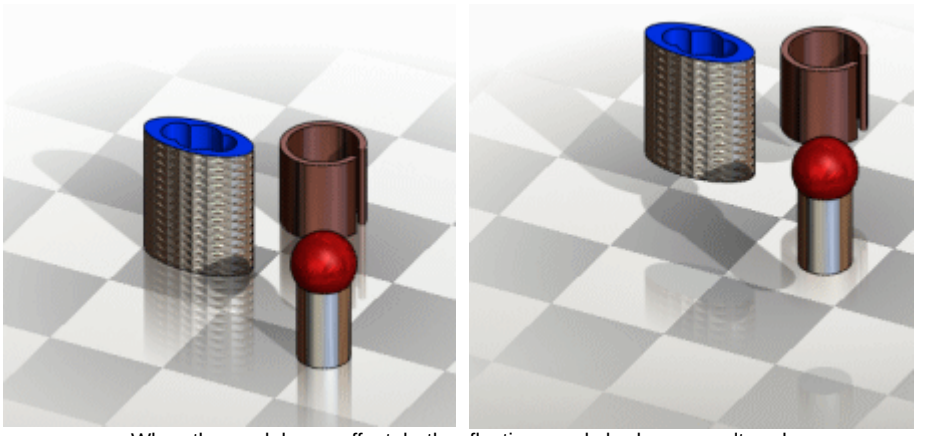
- **Position by selection**. Flips the floor about the selected model geometry based on the planar face you select.





- **Flip floor direction** . Makes the ceiling the floor.

- **Offset** . Offsets the model geometry from the scene floor. Set values or drag the pointers.



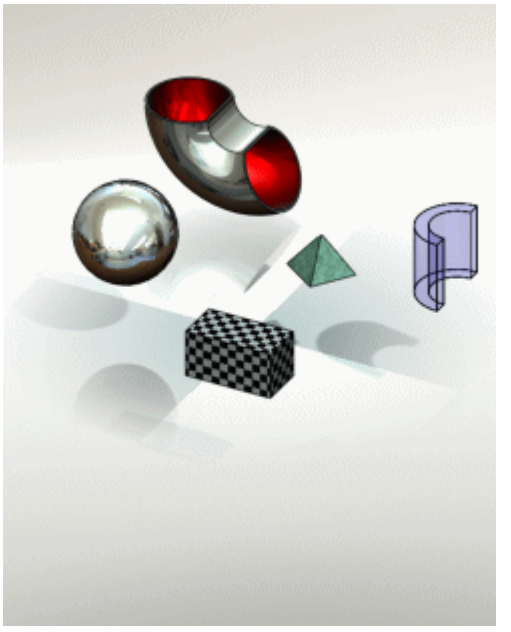
When the models are offset, both reflections and shadows are altered.

- **Rotation**. Spins the floor. Set values or drag the pointers.

The direction of the floorboards and the orientation of the back wall have changed, but the wood grain on the table remains constant.

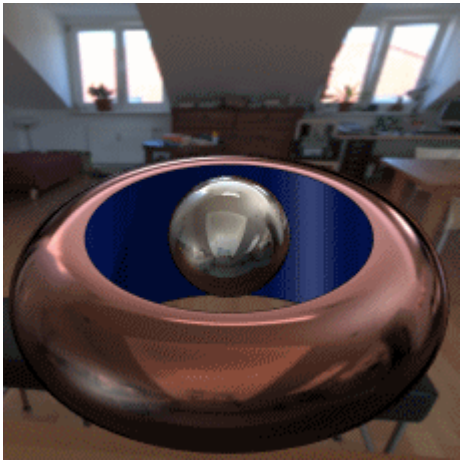
Scale

Controls the size of the environment. Changes in scale are visible with scenes such as the checker board. To modify scale, clear **Auto resize** and set values for **Width**, **Depth**, and **Height**, or drag the pointers.



Environment Rotation

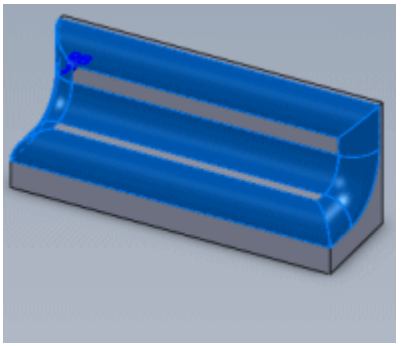
Set a value to rotate the background image about an axis that is normal to the floor.



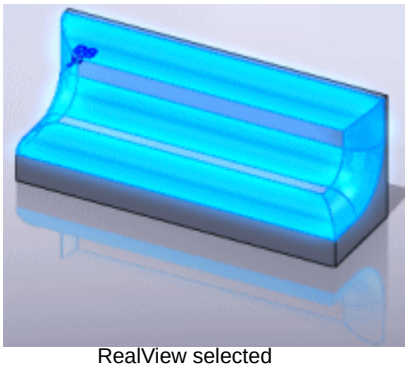
See **Scenes** in the online help.

Dynamic Highlighting

With **RealView**, highlighting for selected edges, faces, or features has changed. Edges glow and appearances are blended.



RealView cleared



RealView selected

3

Sketching

This chapter describes enhancements to sketching in the following areas:

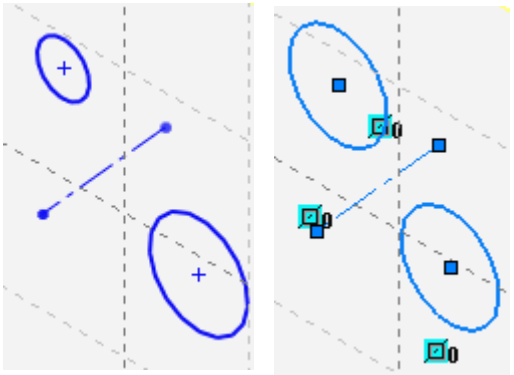
- 3D Sketch Symmetry
- Blocks
- Consolidated PropertyManagers
- Auto Trace Tools
- Show/Hide Sketch
- Sketching in Instant3D
- SketchXpert
- Splines

3D Sketch Symmetry

Symmetry about a line is available in 3D sketches on a plane when all entities are on the plane. Mirror and dynamic mirror are also available in this mode. The symmetry constraint between two entities and a plane is available in 3D sketching.

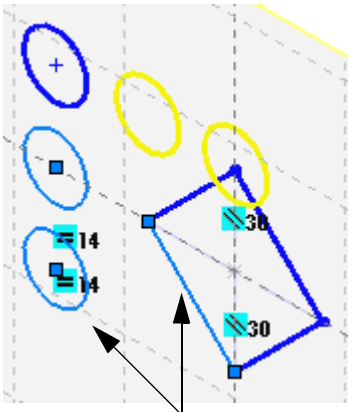
With 2D sketches created on 3D sketch planes, you can:

- Add **Symmetric** relations.

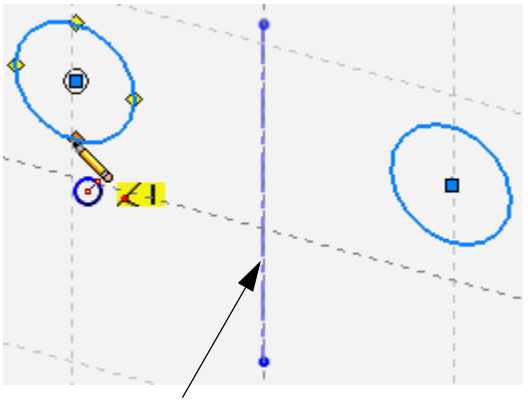


To add symmetry between the 2 arcs, you also need to also select a third entity (the centerline).

- Use the **Mirror** and **Dynamic Mirror** tools.



Mirror: The two lower circles were mirrored about the line.



Dynamic Mirror: The circle was mirrored about the construction line.

Blocks

Create 2D Blocks in 3D Sketches

You can create blocks using 2D sketches on 3D sketch planes. Functionality includes:

- Perform any 2D block command (save, explode, etc.).
- Import blocks saved in 2D into a 3D sketch.
- Combine imported 2D blocks with 2D sketches created on a 3D sketch plane.
- Add relations and dimensions.

See **Layout-based Assembly Design** on page 6-7.

See **Layout Based Assembly Design** in the help.

Area/Hatch Fill

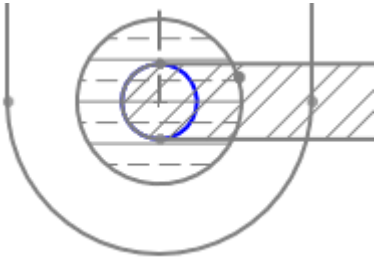
You can add **Area/Hatch Fill** patterns to 2D sketches, 2D sketches on 3D sketch planes, and to sketches converted to blocks.

In **Edit Sketch** mode, click **Tools**, **Sketch Tools**, **Area/Hatch Fill** to add area hatch/fill to sketches before or after converting the sketch entities to a block.

The properties of **Area/Hatch Fill** for blocks include:

- Uses a PropertyManager that is similar to drawings. It includes a **Color** option which displays the **Color** palette for **Solid** fill.
- Functions like area/hatch fill in drawings.
- Applies to all block commands. You can manipulate the block, save the area/hatch fill with the block document, and reuse it in other models.

Block color overrides crosshatch color.



Area Hatch/Fill applied to two intersecting blocks.

See **Area Hatch/Fill PropertyManager** in the help.

Consolidated PropertyManagers

Rectangle-based tools use a new, consolidated **Rectangle** PropertyManager. The consolidated **Arc** and **Circle** PropertyManagers were redesigned to match the **Rectangle** PropertyManager's interface.

New rectangle tools (Sketch toolbar) include:

- **Center Rectangle**
- **3 Point Corner Rectangle**
- **3 Point Center Rectangle**

Rectangle-based tools in the **Rectangle** PropertyManager include:

Rectangle Type		
• **Corner Rectangle**		Sketches standard rectangles.
• **Center Rectangle**		Sketches rectangles at a center point.
• **3 Point Corner Rectangle**		Sketches rectangles at a selected angle.
• **3 Point Center Rectangle**		Sketches rectangles with a center point at a selected angle.
• **Parallelogram**		Sketches standard parallelograms.

You can select any tool from the **Arc**, **Rectangle**, or **Circle** flyout tools, and change tools from the relevant PropertyManager.

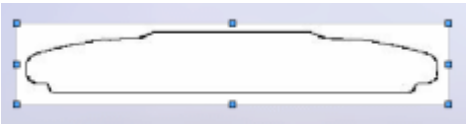
See **Rectangles** in the help.

Auto Trace Tools

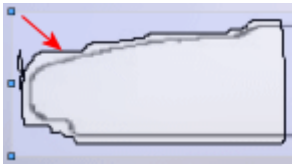
This tool can help convert raster data to vector data. In **Tools**, **Sketch Tools**, **Sketch Picture**, open a document and click to select conversion options. Options include:

- **Trace Settings**
- **Display Options**
- **Adjustments**

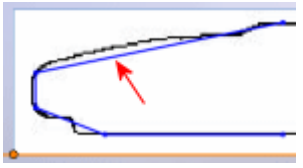
Once you convert the document to vector data, you have a sketch that you can modify, save, and use as the basis for a 3D model.



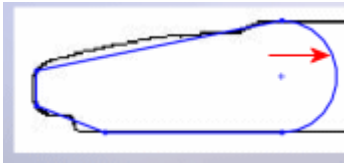
1. Open raster data.



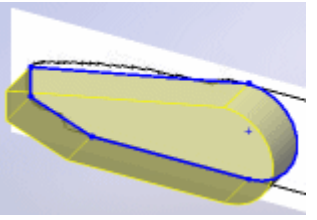
2. Trace shape outline.



3. Convert raster data to vector data.



4. Modify sketch.



5. Create 3D model.

See **Sketch Picture** in the help.

Show/Hide Sketch

From the shortcut menu, you can edit an absorbed 2D sketch and show or hide another absorbed 2D or 3D sketch in the same feature to reference it. This functionality allows you to show or hide an absorbed sketch while editing another absorbed sketch that is part of the same feature.

Apply this functionality to features such as:

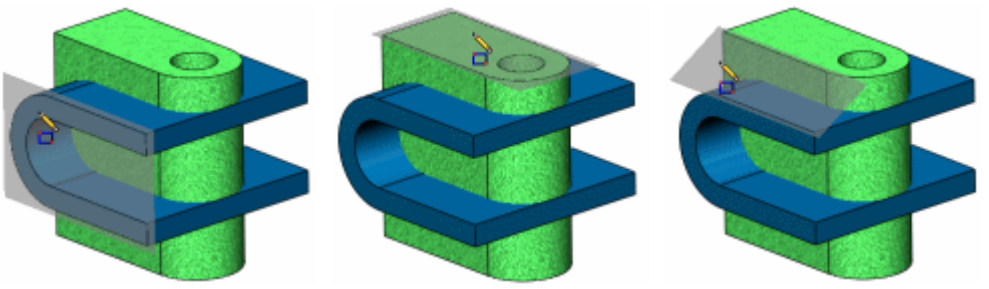
- Lofts with three or more profiles and two guide curves.
- Sweeps with two or more guide curves.
- Sheet metal parts in which you select multiple edges in an edge flange feature.
- Models that include three or more holes created with the Hole Wizard.

Sketching in Instant3D

You can streamline your workflow from sketch to features.

For sketching:

- Use any sketch tool to highlight and activate a planar face or plane.
- Start to sketch on the selected entity, or move to any another planar face or plane to activate it.

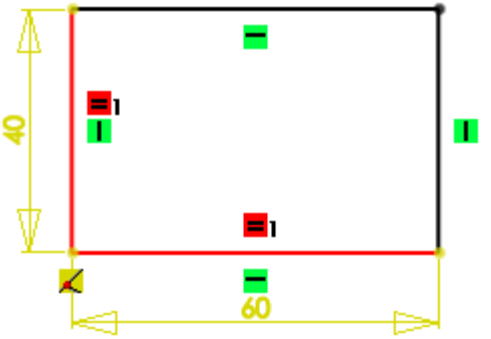


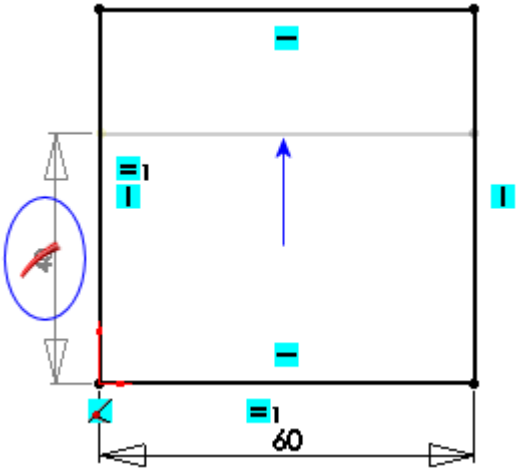
See **Instant3D** on page 4-6.

SketchXpert

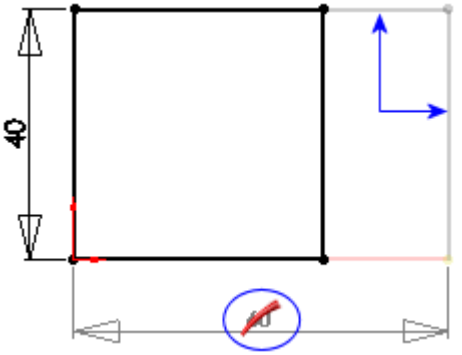
Enhancements

- Displays images of old geometry when solutions cause that geometry to move.
- Displays dimensions and relations to be deleted with a strikethrough.
- Operates with 3D sketches.
- Generates through solutions faster.

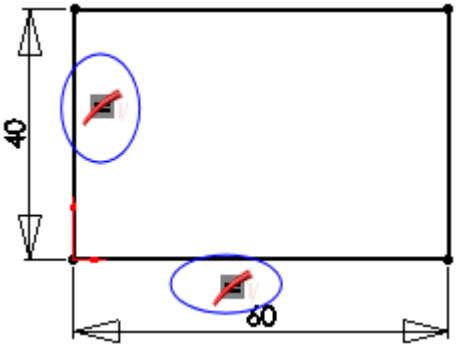




Solution 1: The dimension (40) is deleted, and SketchXpert displays the location of the old geometry.



Solution 2: The dimension (60) is deleted, and SketchXpert displays the location of the old geometry.



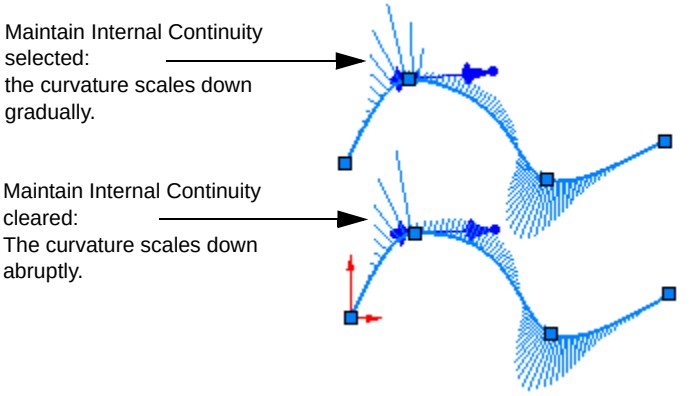
Solution 3: The equal relations are deleted.

See **Resolving Over Defined Sketches** in the help.

Splines

Continuity at Handles

You can manipulate spline handles and maintain the splines' internal curvature when you select the option **Maintain Internal Continuity** in the **Spline** PropertyManager.



Continuity at handles is on by default for newly sketched splines.

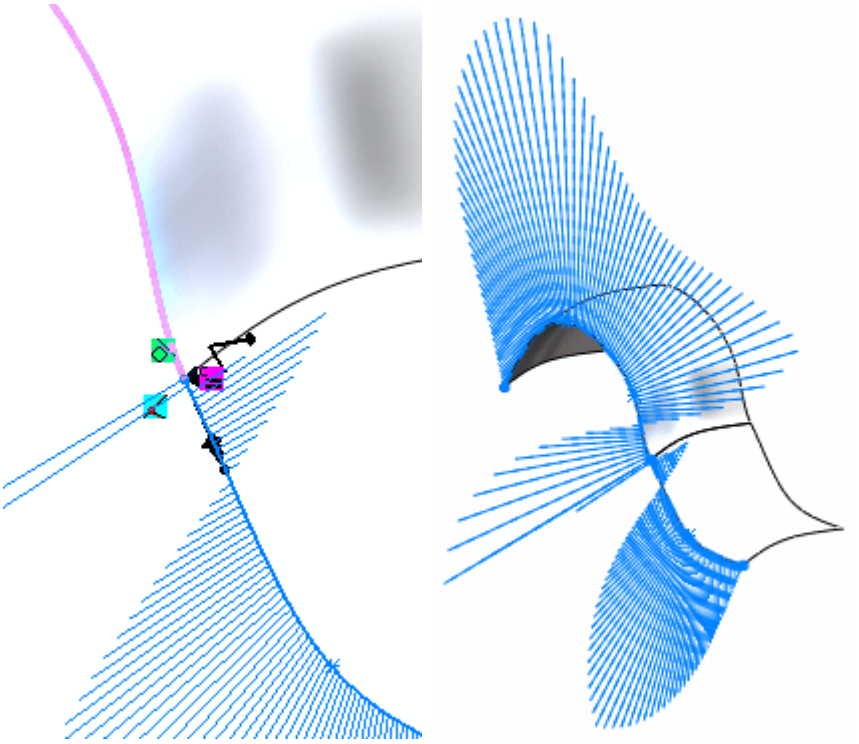
Curvature Constraints

Both the radius of curvature and the vector (direction) match when you add an **Equal Curvature** relation between adjacent splines. This creates splines that are curvature continuous at the boundary. Curvature continuous sketches produce smoother surfaces with tools such as the boundary surface feature.

See **Boundary Surfaces** on page 4-2.

Requirements:

- The second spline must unconstrained.
- The two splines must share an endpoint.

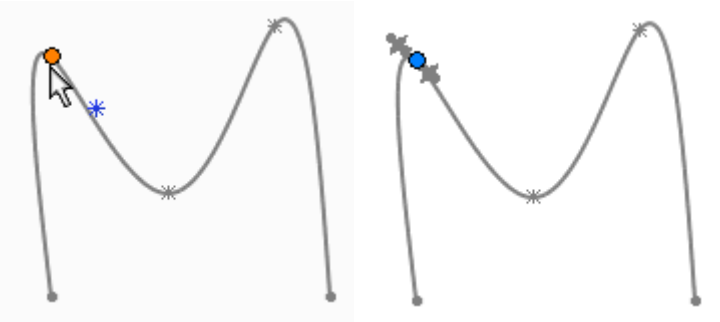


The value of the 2nd derivative is zero on both curves.

Spline Manipulators Available while Not Editing a Sketch

When you sketch a spline and exit the sketch, you can select a spline or spline point to display:

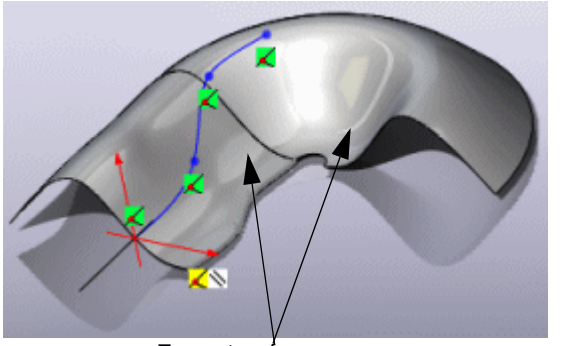
- Active spline handles
- Spline polygon
- Spline handle at the selected point



Previously, you needed to be in **Edit Sketch** mode to display the spline manipulators.

Spline on Surface

When you sketch a spline across multiple surfaces (**Spline on Surface** tool), the surfaces must be tangent. The surfaces can be combinations of surfaces and surfaces on solids.



4

Features

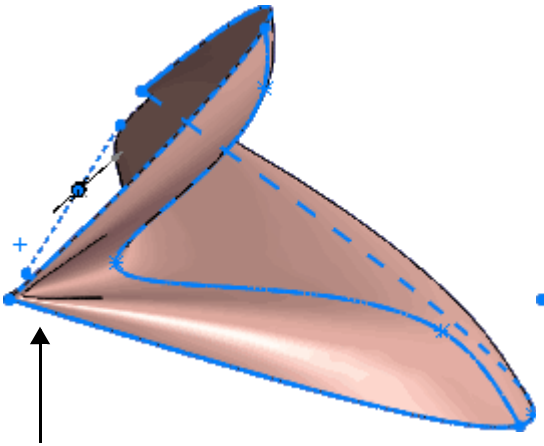
This chapter describes enhancements to Features in the following areas:

- Boundary Surfaces
- Fillets
- Hole Series
- Hole Wizard
- Instant3D
- Patterns
- Split Lines and Parts
- Sweeps

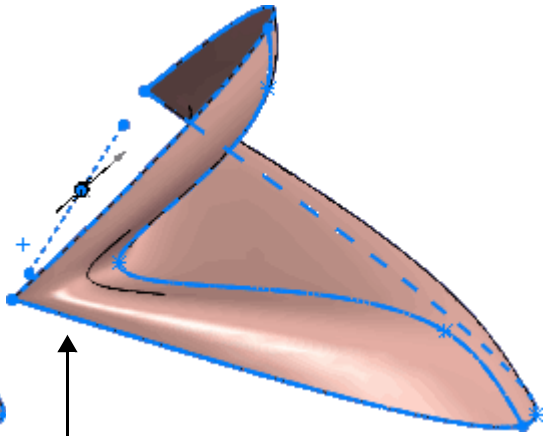
Boundary Surfaces

Linear Option

The new **Linear** option (under **Dir1** or **Dir2 curves influence**) extends the influence of the curve linearly over the entire boundary surface, similar to a ruled surface. This option helps to avoid excessive curvature (pocketing) effects. Pocketing occurs with highly-indented guide curves on surfaces where curves in a single direction are coincident to each other.



Global option



Linear option

Tangent Influence

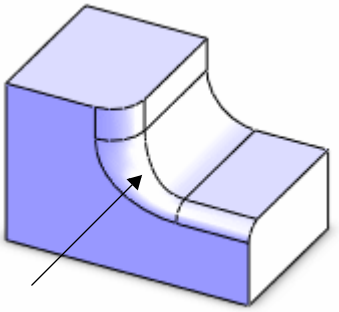
The **Tangent Influence** option has been renamed to **Tangent influence %**. It no longer appears under the **Dir1** or **Dir2 curves influence** list. **Tangent influence %** is a separate option under **Direction 1** or **2** and has a slider to set the amount of influence.

See **Splines** on page 3-8.

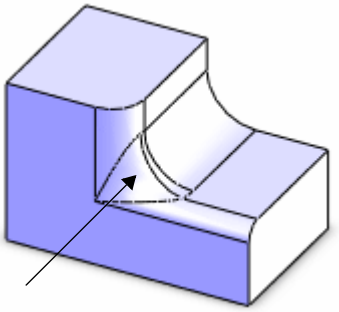
Fillets

Fillet Corners

Use the new **CornerXpert** (**FilletXpert** PropertyManager) to create and manage fillet corner features where exactly three filleted edges meet at one vertex.



Original fillet corner



Alternative fillet corner

You can:

- Choose alternative fillet corners
- Copy a fillet corner to other compatible fillet corners

See **FilletXpert** in the help.

Fillet Selection

When adding or changing fillets using the FilletXpert, selecting an individual edge or fillet displays a context toolbar to help you select multiple edges or fillets.

Hover over a toolbar icon to highlight entities in the graphics area. Click the toolbar icon to select the appropriate entities and populate the PropertyManager.

In the toolbar tooltips:

- **Right** and **Left** = the right or left face of the entity.
- **Start** and **End** = the start or end vertex of the entity.
- **Virtual** = the adjacent tangent entities that the software treats as one entity.

Hole Series

Enhancements:

Tabs

The **Hole Series** PropertyManagers no longer work in a linear fashion. Click tabs in any order to access each PropertyManager's information. All tabs are always visible. There is a new Smart Fasteners tab available if you install SolidWorks Toolbox.

Smart Fastener tab

Autosizing

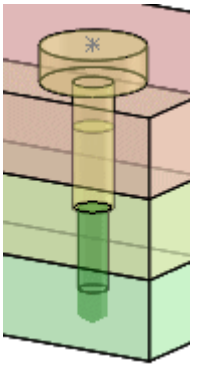
If you size a Hole Series, the software automatically sizes related Smart Fasteners. If you modify the size of the Hole Series, the software adjusts the Smart Fastener size.

You must install and activate SolidWorks Toolbox.

To turn on automatic sizing, select **Auto-size based on start hole** on the Smart Fasteners tab.

See **Smart Fasteners** on page 6-21.

Previews



Hole Series uses improved previews that specify individual components of the Hole Series. Individual Smart Fasteners components also highlight.

For example, when focus is on the **End Hole** tab, the end hole highlights in the graphics area.

See **Hole Series** the help.

Hole Wizard

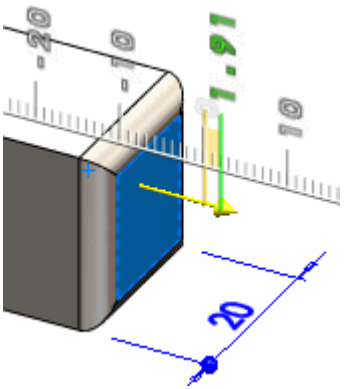
Enhancements

- The **Custom Size** group box has been removed. Click the new Show **custom sizing** option under **Hole Specifications** to set custom sizing information.
- When your model has configurations requiring different Hole Wizard hole sizes, you can specify the configurations to modify using the PropertyManager or a design table.

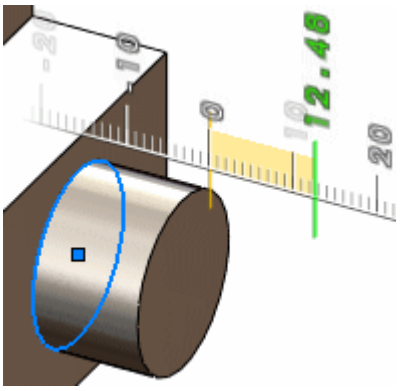
See **Hole Wizard PropertyManager - Type Tab** in the help.

Instant3D

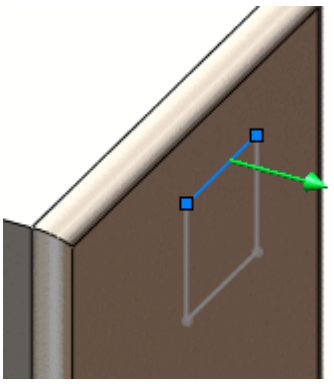
Instant3D allows you to do the following:



- Drag geometry and dimension manipulators to resize features.

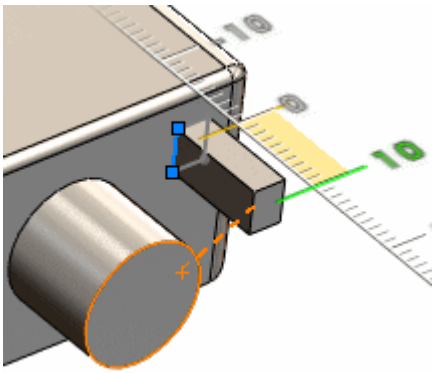


- Use on-screen rulers to precisely measure modifications.

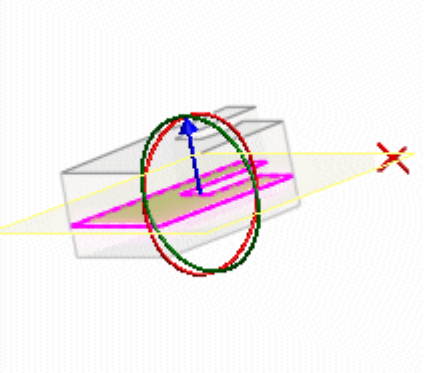


- Create extruded bosses and cuts from selected contour or sketches.

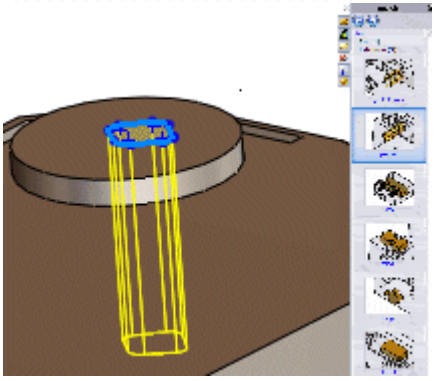
- Snap to geometry using drag handles.



- Dynamically section model geometry to view and manipulate features.



- Use 3D clip art of various model content, such as features, sketches, tables, blocks, views, etc., to search through models.



See **SolidWorks Search and Help** on page 1-2, **Sketching in Instant3D** on page 3-6, and **Dissect Files** on page 13-21.

See **Instant3D** in the help.

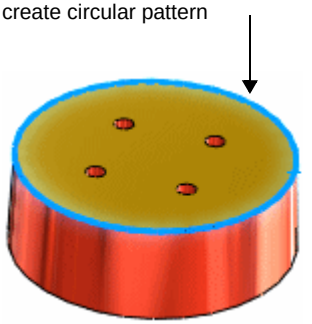
Patterns

Circular Patterns

You can select these entities for **Pattern Axis** in the PropertyManager:

- Cylindrical face or surface
- Revolved face or surface
- Circular edge or sketch line

Circular edge used as Pattern Axis to create circular pattern



Cosmetic Patterns

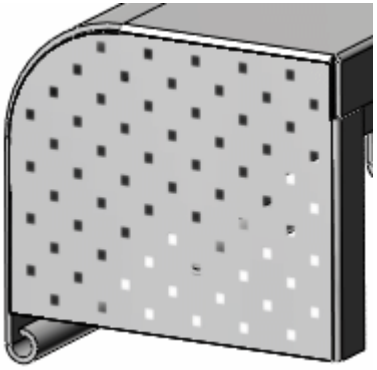
The new cosmetic pattern feature allows you to cosmetically define and display patterns of holes instead of showing a fully-tessalated solid model. This feature reduces rebuild time by not creating the pattern geometry.

- You can apply cosmetic patterns only to planar, parallel faces.
- Drawings of cosmetic patterns show a simplified representation of the pattern.

RealView Graphics must be enabled to create cosmetic patterns. Click **View**, **Display** and select **RealView Graphics**.

To create cosmetic patterns:

1. From the Task Pane, select the **RealView** tab, click **Miscellaneous**, **Pattern**, then double-click **cosmetic hole pattern** or drag it onto the model.
2. Select a face for **Fill Boundary**.
3. Under **Pattern Layout**:
   a) Select a layout.
   b) Set the spacing options.
   c) Select an edge to determine the pattern direction.
4. Under **Seed Types**:
   a) Select a seed type.
   b) Set the size options.
5. Click .



See **Cosmetic Pattern PropertyManager** in the help.

Split Lines and Parts

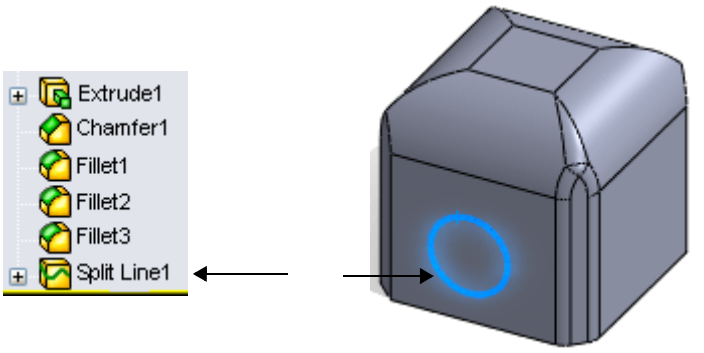
Split Lines

When you create split lines, unchanged edges are reusable in downstream features and changed edges update.

Supported features:

- Chamfers
- Drafts
- Fillets
- Shells

For example, in this model you insert a split line on the front face as the last item in the FeatureManager design tree. If you reorder **Split Line1** to follow the **Extrude1** feature, the model retains the downstream chamfer and fillets. Previously, reordering was unsuccessful.



Split Parts

Enhancements:

- You can reattach a derived part to a specified stock part, split feature, or body.
- When you change the split feature geometry, no new derived parts are created. The existing derived parts are updated, preserving parent-child relations.
- When you split parts, you can select **Copy custom properties to new parts** in the **Split** PropertyManager.

See **Split and Save Bodies PropertyManager** in the help.

Sweeps

You can sweep a solid body along a path to cut 3D material using the new **Solid sweep** option. The most common usage is in creating cuts around cylindrical bodies. This option would also be useful for end mill simulation.

Requirements:

1 The solid body must:
   - Be a revolve.
   - Contain only analytical geometry.
   - Not be merged with the model.

2 The path must be tangent within itself (no sharp corners) and must begin at a point on or within the solid body profile.

To create a solid body swept cut:

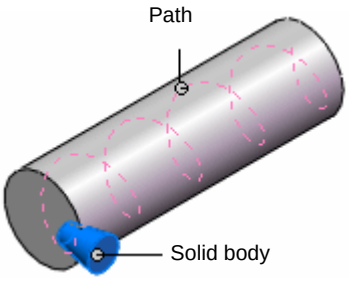
1 Open **Sweep.sldprt**.

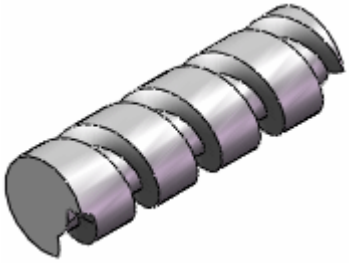
2 Click **Swept Cut** (Features toolbar) or **Insert**, **Cut**, **Sweep**.

3 Under **Profile and Path**:
   a) Select **Solid sweep**.
   b) Select the revolve for **Tool body** .
   c) Select **Helix/Spiral1** for **Path** .

4 Click .

5 Hide the revolve solid body.





5

Parts

This chapter describes enhancements to parts in the following areas:

- Mate to Coordinate Systems
- Inserting Sketches when Inserting Parts
- Breaking the Link to a Part
- Custom Properties of Parts
- Automatic Positioning of Parts Using Mate References
- Isolating Bodies in Part Mode

Mate to Coordinate Systems

While inserting a body into a part, you can apply a coincident mate between a coordinate system of the inserted body and a coordinate system in the target part. If you also select to align the axes, the one coincident mate fully constrains the body.

This is useful in designs with modular parts whose positions and orientations are fully defined.

Inserting and Mirroring Parts

When inserting parts and deriving parts by mirroring, you can now:

- Insert solid bodies, sketches and absorbed sketches
- Break the link to the original part
- Retain the custom properties of the inserted part
- Position parts automatically using mate references

See **Insert Part** and **Mirror Part** in the help.

Inserting Sketches when Inserting Parts

When you insert a part into another part, you can include solid bodies and non-absorbed and absorbed sketches. Inserted sketches are linked to the sketch from the inserted part file and remain linked until the inserted part is unlinked.

All transformed elements that came with the inserted part are placed into folders located under the inserted part feature icon.

Breaking the Link to a Part

You can break the link to a mirrored or derived part and still keep the original features so that they can be edited in the new part. This facilitates the design of left and right side parts that vary slightly in their definitions.

You can break the link to the part:

- When you insert the part
- By clicking **Break All** in the External References dialog box

Custom Properties of Parts

You can retain the custom properties of a part when you:

- Insert the part into another part

  If the target part file already has custom properties of the same name, then those original custom properties take priority over the inserted part's custom properties.

- Create a mirrored part using **Insert, Mirrored Part**

  When you create a mirrored part, the mirrored part's custom properties table is linked to the table of the original part. You can unlink any individual custom property of the mirrored part by editing its value field.

Automatic Positioning of Parts Using Mate References

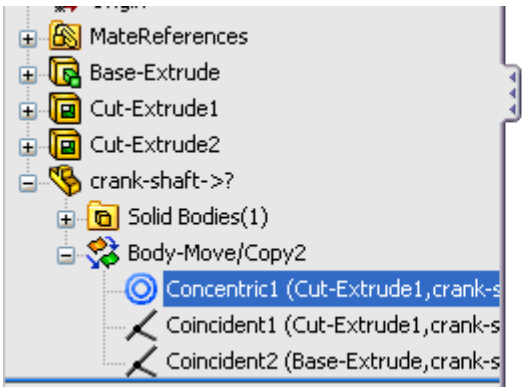
When you insert a part into an existing part file, any existing mate references in the inserted part are used automatically to place the inserted part body and the body move constraint is added to the feature tree.

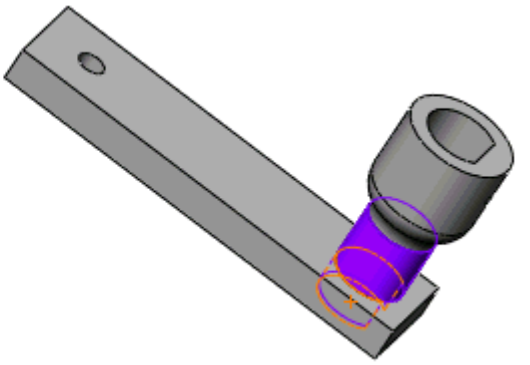
This feature applies to:

- All selectable reference entities
- All mate reference types (default, tangent, coincident, concentric, and parallel)
- All mate reference alignments (any, aligned, anti-aligned, closest)

A preview shows the application of the existing mate reference as the part is inserted and before you OK the location.

If you select the body move constraint in the FeatureManager design tree, it is highlighted in the graphics area.





Hands-on Example

See **Mates in Multibody Parts** in the help.

Isolating Bodies in Part Mode

You can isolate one or more selected bodies in a multibody part so that only the isolated bodies are fully displayed. The Isolate dialog box lets you control whether the bodies that are not isolated are hidden or transparent. The display you select applies to a single view or multiple views.

Exiting Isolate resets the display of all bodies to the display that was set prior to using isolate.

This capability is useful to product designers that work in multibody part mode to make molds or weldments.

See **Isolate** in the help.

6

Assemblies

This chapter describes enhancements to assemblies in the following areas:

- General
- AssemblyXpert
- Derived Components
- Hole Alignment
- Layout-based Assembly Design
- Mates
- Selection
- Simplified Representations
- Smart Fasteners

General

Assembly Statistics

Information previously found in **Assembly Statistics** is now reported in the **AssemblyXpert**. See **AssemblyXpert** on page 6-2.

References

Dialog boxes related to references have been updated.

See these topics in the help:

- **Edit Referenced File Locations**
- **Save As with References**
- **Find References**

Stack-Up Analysis

You can analyze tolerance stack-ups in assemblies. See **TolAnalyst** on page 14-17.

Display States in eDrawings®

When you save an assembly as an eDrawings file, the assembly's display states are saved in the eDrawings file. See **SolidWorks Display States** on page 12-12.

AssemblyXpert

The **AssemblyXpert** analyzes performance of assemblies and suggests possible actions you can take to improve performance. This is useful when you work with large and complex assemblies. In some cases, you can select to implement the suggested changes automatically.

Although the conditions identified by **AssemblyXpert** can degrade assembly performance, they are *not* errors. It is important that you weigh recommendations of the **AssemblyXpert** against your design intent. In some cases, implementing the recommendation would improve assembly performance, but would compromise your design intent.

To analyze performance of an assembly:

In an assembly, click **Tools**, **AssemblyXpert** .

See **AssemblyXpert Overview** in the help.

Derived Components

Custom Properties of Mirrored Components

When you mirror components, you can select **Copy custom properties to new components** in the **Mirror Components** PropertyManager.

Derived Component Patterns

You can derive component patterns from the following additional types of feature patterns:

- Curve driven
- Fill

Instances to Skip is now supported in the following additional types of feature patterns:

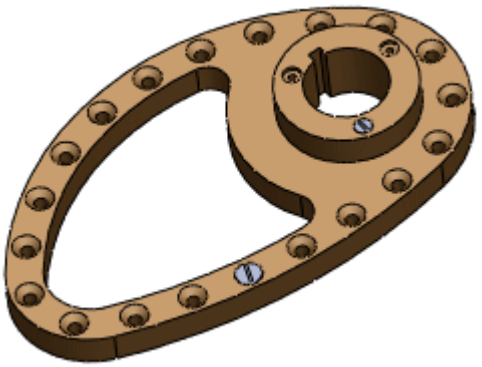
- Sketch driven
- Table driven
- Curve driven
- Fill
- Hole Wizard holes
- Hole Series holes

Additionally, for all types of derived component patterns, you can:

- Propagate component level visual properties.
- Select the parent seed location for patterning the derived component.

To insert components based on a curve-driven feature pattern:

1 Open **Assemblies\patterns\frame_assembly.sldasm**.

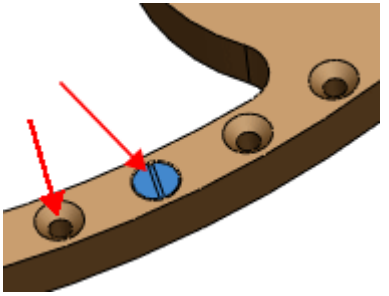


2 On the **Assemble** tab in the CommandManager, expand the **Linear Component Pattern** flyout button and select **Feature Driven Component Pattern**.

3 In the PropertyManager:

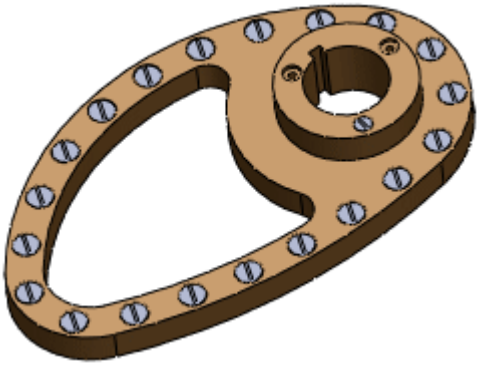
a) For **Components to Pattern**, select the larger screw in the graphics area.

b) For **Driving Feature**, select any hole in the curve-driven feature pattern.



4 Click ✔.

5 The instances of the screw component are added along the curve-driven hole pattern.



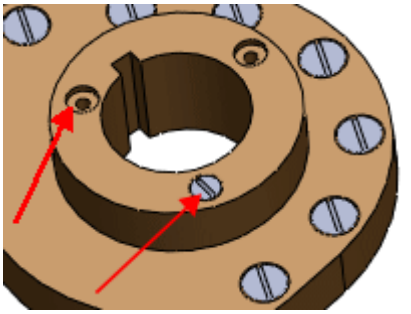
To change the seed position:

1 On the **Assemble** tab in the CommandManager, expand the **Linear Component Pattern** flyout button and select **Feature Driven Component Pattern**.

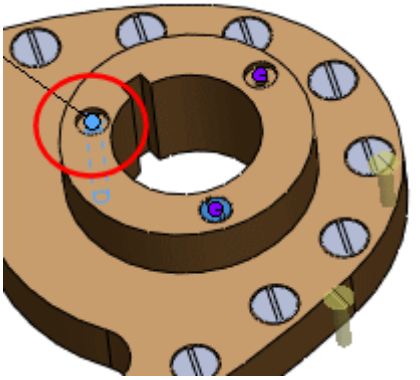
2 In the PropertyManager:

a) For **Components to Pattern**, select the smaller screw in the graphics area.

b) For **Driving Feature**, select the hole as shown.

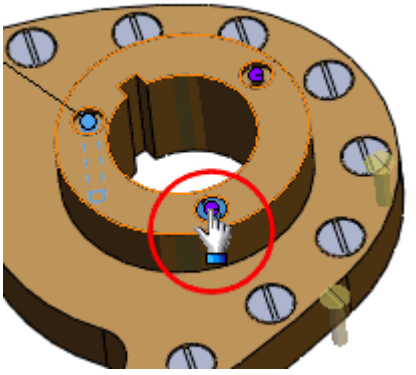


The hole you select becomes the seed feature.



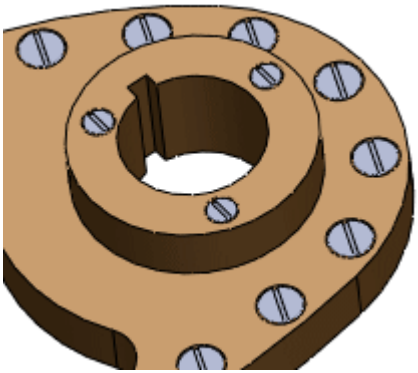
3 In the PropertyManager, under **Driving Feature**, click **Select Seed Position**. Possible seed positions highlight in the graphics area.

4 Select the seed position that corresponds with the screw you selected to pattern.



The screws align with the hole pattern.

5 Click ✔.

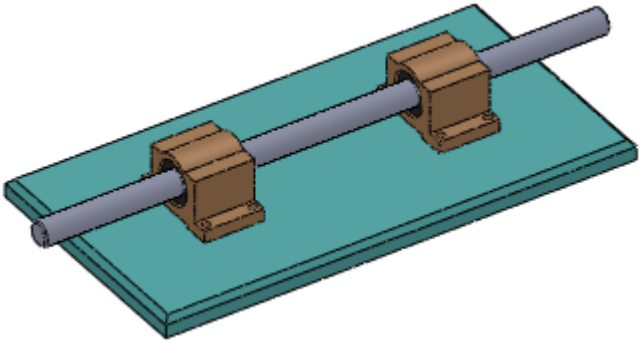


Hole Alignment

You can check an assembly for misaligned holes.

To find misaligned holes:

1 Open **Assemblies\alignment\shaft_assembly.sldasm**.



2 Click **Hole Alignment** (Assembly toolbar) or **Tools**, **Hole Alignment**.

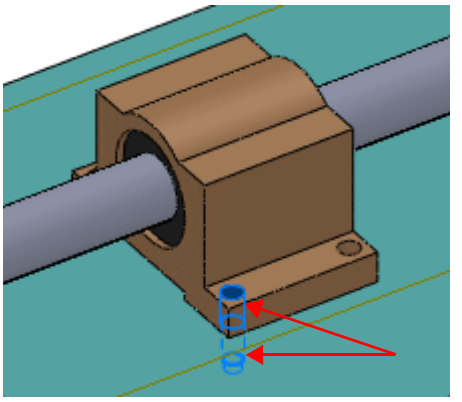
3 In the PropertyManager, set **Hole center deviation** to **1.00mm**.

4 Click **Calculate**.

Pairs of holes whose centers are within 1.00mm of each other, but not aligned, are listed under **Results**. The maximum deviation between centers is listed.

5 Select an item under **Results**.

The pair of holes highlights in the graphics area.



6 Expand an item under **Results**.

The two holes in the misaligned pair are listed.

See **Hole Alignment** in the help.

Layout-based Assembly Design

Enhancements have been made that provide a layout-based assembly design environment where you can switch back and forth between top-down and bottom-up design methods. You can create, edit, and delete parts and blocks at any point in the design cycle without any history-based restrictions. This is particularly useful during the conceptual design process, when you frequently experiment with and make changes to the assembly structure and components.

Virtual Components

When you create components in the context of an assembly, the software now saves them inside the assembly file, and you can immediately begin modeling. Later, you can save the components to external files or delete them.

In addition to streamlining the workflow, other advantages include:

- You can rename these virtual components in the FeatureManager design tree, avoiding the need to open, save as a copy, and use the **Replace Components** command.
- You can make one instance of a virtual component independent of other instances in a single step.
- The folder where you store your assembly is not cluttered with unused part and assembly files resulting from iterations of component designs.

To create a virtual component:

1. In an assembly, click **New Part** (Assembly toolbar) or **Insert**, **Component**, **New Part**.

   A new part appears in the FeatureManager design tree as **[Part*n*]**. The square brackets indicate that it is a virtual component.

2. Select a plane or planar face on which to position the new part.

   The editing focus changes from the assembly to the new part. An **InPlace** mate is added between the plane you selected and the front plane of the new part, and a sketch opens.

3. Construct the part features, using the same techniques you use to build a part on its own. Reference the geometry of other components in the assembly as needed.

4. To return to editing the assembly, click to clear **Edit Component** (Assembly toolbar).

You can also create virtual components from blocks in layout sketches in assemblies. See **Layout Sketches** on page 6-8.

To save a virtual component to an external file:

1. Right-click the part and select **Save Part(in External File)**.
2. In the **Save As** dialog box, select the part in the **File Name** list.
3. Click **Same As Assembly** or **Specify Path**.
4. Click **OK**.

Layout Sketches

Blocks have been enhanced so you can mix layout-based design with assembly design. Using this new assembly layout technique, you can:

- Create a new type of sketch, called a **Layout**, in an assembly.
- Insert a layout in an existing assembly.
- Constrain blocks in the layout to components in the assembly, and vice versa.
- Create parts from blocks in a layout. These parts are:
  - Constrained to the blocks so they do not violate the layout, but are otherwise free for you to constrain.
  - Contain instances of the blocks. You can edit the blocks in either the layout or the parts.
  - Created as virtual parts, to streamline the workflow. See **Virtual Components** on page 6-7.

Hands-on Example

See **Layout-based Assembly Design** in the help.

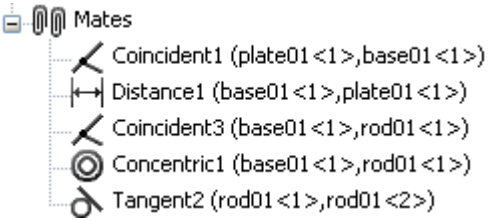
Mates

Mate Icons in the FeatureManager Design Tree

In the FeatureManager design tree:

- New icons indicate the type of mate. (Previously, all mates used .)

  Example:



- When a mate is resolved but missing a reference, the new icon remains the same and the mate is flagged with . (Previously, the icon changed to and was flagged with .)

See **Mate Icons in the FeatureManager Design Tree** in the help.

Mating to Origins and Coordinate Systems

You can apply a coincident mate between the origin or a coordinate system of a component and the origin or a coordinate system of:

- The assembly.
- Another component.

If you also select to align the axes, the one coincident mate fully constrains the component.

This is useful in designs with modular components whose positions and orientations are fully defined.

See **Mating to Origins and Coordinate Systems** in the help.

Mate PropertyManager

The following have been added to the **Mate** PropertyManager:

- Several new mate types
- **Mechanical Mates** group box
- **Analysis** tab (For COSMOSMotion™ only.)

Standard Mates

The following mate type has been added:

Lock . Maintains the position and orientation between two components. The components are fully constrained relative to each other. A **Lock** mate has the same effect as forming a sub-assembly between the two components and making the sub-assembly rigid. Previously, **Lock** mates were called fixed joints in COSMOSMotion.

See **Lock Mate** in the help.

Advanced Mates

The following mate types have been added:

- **Linear/Linear Coupler** . Establishes a relationship between the translation of one component and the translation of another component. Previously, **Linear/ Linear Coupler** mates were created using a joint coupler between two translational joints in COSMOSMotion.

See **Linear/Linear Coupler Mate** in the help.

- **Path Mate** . Constrains a selected point on a component to a path. You define the path by selecting one or more entities in the assembly. You can define pitch, yaw, and roll of the component as it travels along the path.

See **Path Mate** in the help.

The following mate types have been moved to the new **Mechanical Mates** group box:

- **Cam**
- **Gear**
- **Rack Pinion**

Mechanical Mates

The following mate types have been added:

- **Screw** . Constrains the same degrees of freedom as a concentric mate, with the addition of a pitch relationship between the rotational degree of freedom and the translational degree of freedom along the axis. Translation along the axis causes rotation according to the pitch relationship, and vice versa. Previously, **Screw** mates were called screw joints in COSMOSMotion.

See **Screw Mate** in the help.

- **Universal Joint** .The rotation of one component (the output shaft) about its axis is driven by the rotation of another component (the input shaft) about its axis. Previously, **Universal Joint** mates were called universal joints in COSMOSMotion.

See **Universal Joint Mate** in the help.

Analysis Tab

You can add the following properties to mates for use in COSMOSMotion analysis. (You can add the properties without having COSMOSMotion added in.)

- **Load Bearing Faces**. Associates additional faces with a mate to define which faces share in bearing the load. (Previously, **Load Bearing Faces** were called load references, and were added using the **Load References** PropertyManager.)
- **Friction**. Associates friction properties with a mate. You can specify the coefficient of friction or the materials involved in the mate.
- **Bushing**. Associates bushing properties with a mate. You can specify translational and torsional stiffness, damping, force, and torque.

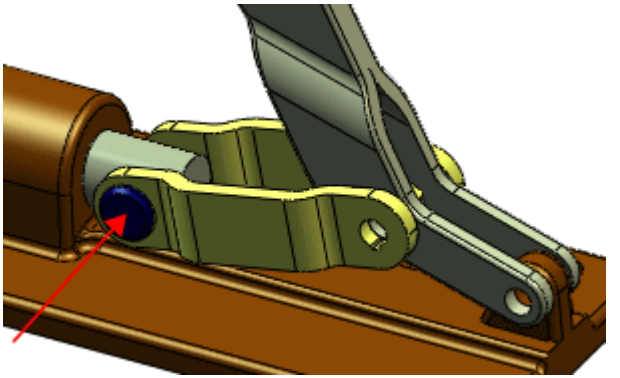
See **Mate Analysis** in the help.

Copy with Mates

When you copy a component, you can also copy its mates.

To copy mates when you copy a component:

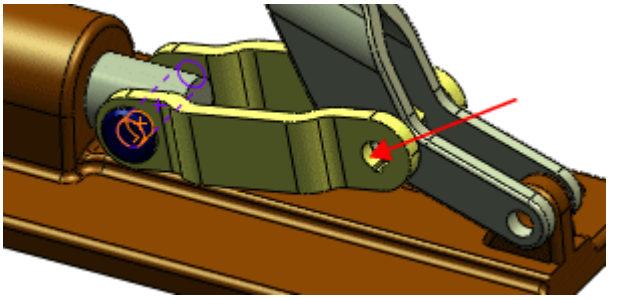
1. Open **Assemblies\fixture\clamping_fixture.sldasm**.
2. Click **Copy with Mates** (Assembly toolbar) or **Insert**, **Component**, **Copy with Mates**.
3. In the PropertyManager, for **Selected Components**, select **pin106** in the graphics area.



Two mates, one concentric and one coincident, are listed under **Mates**.

4. Click in the selection box under the concentric mate.

   The mate reference highlights in the graphics area.
5. Select the hole of **link105** as shown:



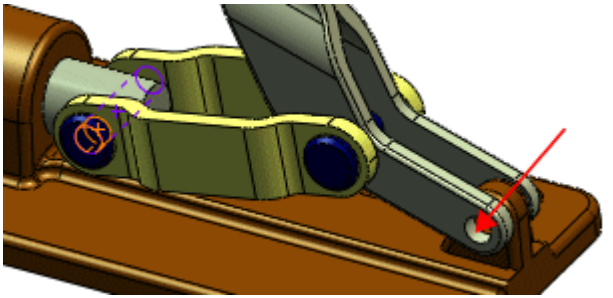
In the graphics area, a preview of the pin appears in the hole, and the mate reference for the next mate highlights. In the PropertyManager, the selection box for the coincident mate highlights.

6 In the PropertyManager, for the coincident mate, select **Repeat** to use the same mate reference as used by the original pin.

7 Click ✔.

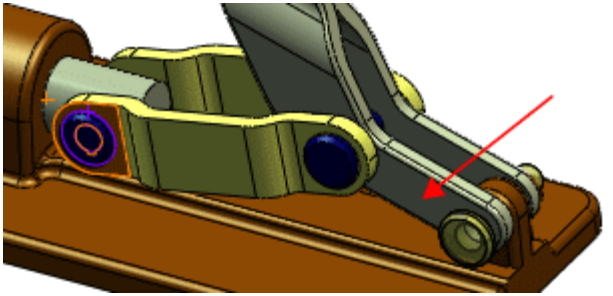
The copy of the pin is added to the assembly. The selection box clears so you can add another copy.

8 Select the hole in the lever.



9 In the PropertyManager, for the coincident mate, clear **Repeat**.

10 Select the flat face of the lever as the mate reference for the coincident mate for this copy of the pin.



11 Click ✔ twice.

The copies of **pin106** appear in the FeatureManager design tree. Their mates appear in the **Mates** folder.

See **Copy with Mates** in the help.

Selection

Selecting Sub-Assemblies in the Graphics Area

You can select a sub-assembly in the graphics area by right-clicking one of its components and choosing **Select Sub Assembly**. If the component is in a nested sub-assembly, a list displays the hierarchy. Move the pointer over the list to highlight the various sub-assemblies, then click the one you want.

Selection Tools

New tools help you to select components in assemblies. Click the down arrow on **Select** (Standard toolbar) and select:

- **Volume Select**
- **Select Suppressed**
- **Select Hidden**
- **Select Mated To**
- **Select Internal Components**

See **Selecting Components** in the help.

Advanced Component Selection

The **Advanced Select** dialog box now uses a grid interface, making it easier to set up and repeat searches. Additionally, the number of searchable criteria has been increased.

Filtering the FeatureManager Design Tree

You can use **Filter** at the top of the FeatureManager design tree to filter items displayed in the tree.

You can filter by:

- Component and feature name
- Component show/hide state
- Tags that you add (see **Tags** on page 1-6)

You can also specify to match the graphics area and the filter results.

To filter by show/hide state:

1. Open **Assemblies\filter\tableassembly.sldasm**.

   In the FeatureManager design tree, note that the components **bracket** and **head** are hidden.

2. In **Filter**, click the down arrow and select **Filter Hidden/Suppressed Components**.

   The components **bracket** and **head** disappear from the FeatureManager design tree.

3. Click the down arrow again and clear **Filter Hidden/Suppressed Components** to make the components reappear.

When you hide components using **Isolate**, you can match the FeatureManager design tree to the graphics area by selecting **Filter Hidden/Suppressed Components** in **Filter**.

To filter the graphics area:

1. In **Filter**:
   a) Click the down arrow and *clear* **Filter Graphics View**.
   b) Type **CLA**.

   All components except clamps disappear from FeatureManager design tree, but remain visible in the graphics area.

2. Click the down arrow again and *select* **Filter Graphics View**.

   In the graphics area, all components except clamps disappear.

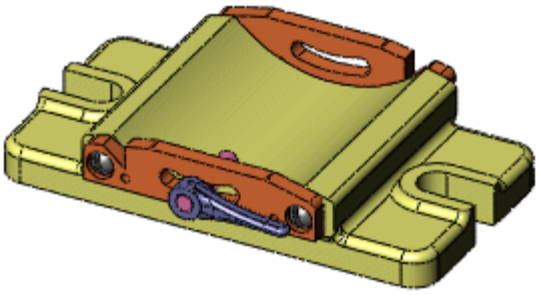
3. Click in **Filter** to clear the filter.

Show Hidden Components

You can toggle the display of hidden and shown components. Then in the graphics area, you can select which hidden components you want to show.

To select hidden components to be shown:

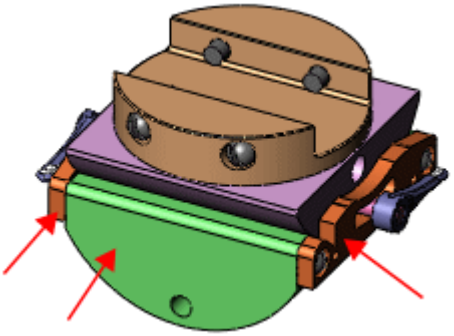
1 Open **Assemblies\vise\ToolVise.sldasm**.



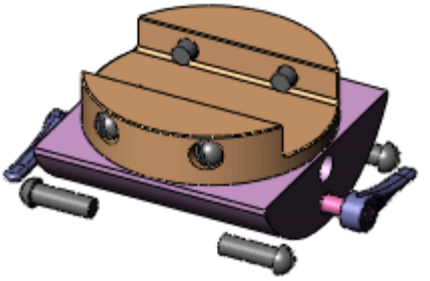
2 Click **Show Hidden Components** (Assembly toolbar).

The **Show Hidden** dialog box appears. In the graphics area, hidden components are displayed and shown components disappear.

3 In the graphics area, select **compound center member**, **upper plate<1>**, and **upper plate<2>** as shown.

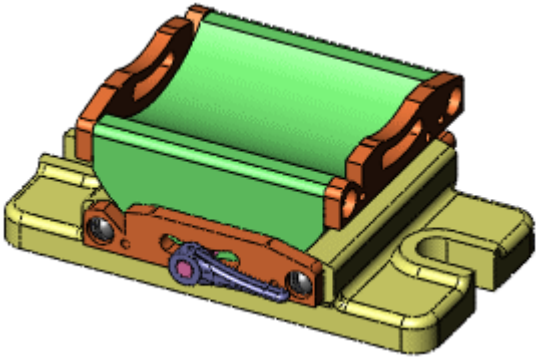


The components you select disappear.



4 In the dialog box, click **Exit Show-Hidden**.

The three components you selected now appear with the other components that were originally shown.



Simplified Representations for Assemblies

Overview

When you want to work on a small subset of components in a large assembly, you can improve assembly performance by opening a simplified representation of the assembly. You specify which components to load; other components are not loaded and not visible, but the effects of their mates are retained.

You specify which components to load by opening the assembly through the **Open** dialog box. While opening, you can select:

- Individual components. (You do not need to fully open the assembly first.)
- A Display State where you previously defined the show/hide state of the components.

Display States

To support simplified representations, you can now make Display States independent from configurations, so all Display States are available in all configurations. The active Display State is no longer remembered per configuration unless you specifically link it.

On the ConfigurationManager tab , Configurations appear at the top of the pane, and Display States appear at the bottom.

You can:

- Right-click a Display State name to open the **Display State Properties** PropertyManager, where you can rename the Display State and select options.
- Select a Display State from the **Open** dialog box.
- Switch Display States using the new Display States toolbar:

- Link a Display State to the active configuration by selecting **Link Display States to Configurations** at the bottom of the ConfigurationManager .

See **Legacy Behavior** on page 6-20 for information on behavior of Display States when you open models created in SolidWorks 2007 or earlier.

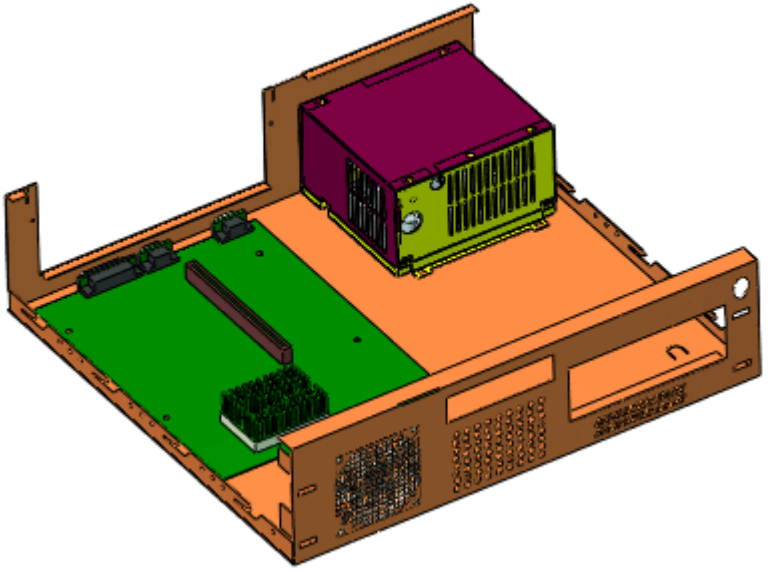
Selective Loading of Components

To load only selected components when opening an assembly:

1. Click **Open** (Standard toolbar) or **File**, **Open**.
2. In the **Open** dialog box:
   a) Browse to **Assemblies\computer\computer.sldasm**.
   b) Select **Quick view / Selective open**.
   c) Click **Open**.

   The following appear:

   - The **Selective Open** dialog box.
   - A simplified FeatureManager design tree showing only components.
   - A preview of the assembly.



3 In the FeatureManager design tree:

a) Expand **Power Supply Assembly-1**.

b) Hold down **Ctrl** and select these three components:

- **Computer Chassis-1**
- **Chassis-1**
- **AC Connector-1**

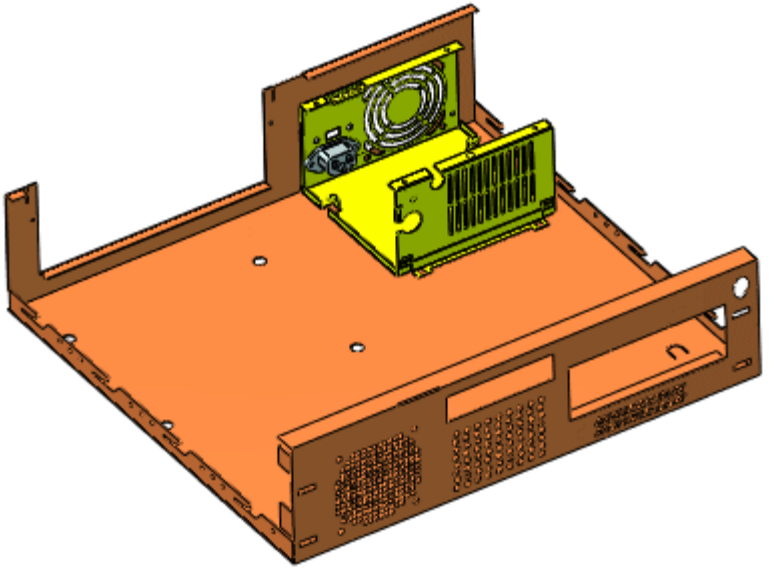
You can also select components in the graphics area.

4 In the **Selective Open** dialog box:

a) Choose **Selected Components**.

b) Click **Open Selected**.

The assembly opens with all components hidden except the three you selected. The hidden components are not loaded into memory. A new display state appears on the ConfigurationManager tab under **Global Display State**.



Legacy Behavior

In assemblies created in SolidWorks 2007 or earlier, each configuration has unique Display States, although the Display States might have the same name (such as **Display State-1**). When you convert an assembly to SolidWorks 2008, the software:

- Assigns a unique name to each Display State, in the format *<configuration_name>_<display_state_name>*.
- Links the Display State to the configuration.

For example, a Display State named **Display State-1** in a configuration named **Default** is renamed to **Default_Display State-1**, and **Link Display State to Configurations** is selected by default.

Smart Fasteners

The **Smart Fasteners** PropertyManager has been updated to include direct controls for adding top and bottom stack components as well as accessing all fastener properties. In the PropertyManager:

- **Results**. Lists the groups of fasteners you are adding or editing.
  - Select a group to view and make changes to its fastener type and properties.
  - Click **Edit Grouping** to view a fastener tree where you can drag items from one group to another.
- **Series Components**. Displays the fastener type for the item that you select in the **Results** list. You can edit the fastener type and add stack components. You can also set an option to automatically update the hardware size whenever the hole diameter changes.
- **Properties**. Displays the properties of the hardware that is selected in **Series Components**. You can edit properties to change the hardware.

The following FeatureManager icons are related to Smart Fasteners:

Smart Fasteners feature. Expand the feature to see items such as components and patterns for the Smart Fastener. Right-click the feature and select **Edit Smart Fastener** to make changes.

Toolbox component.

Toolbox component set to autosize when the mating geometry changes.

Concentric mate associated with a Smart Fastener set to autosize when the mating geometry changes.

You can now access Smart Fasteners and their stack components from the **Fasteners** tab in the **Hole Series** PropertyManager. (The **Fasteners** tab is only available when Toolbox is added in.)

See **Smart Fasteners PropertyManager** in the help.

7

Configurations

This chapter describes enhancements to configurations in the following areas:

- General
- Creating a PropertyManager to Configure Components
- Creating and Modifying Configurations

General

Design Table Feature

The **Design Table** feature now appears in the ConfigurationManager instead of the FeatureManager.

Hole Wizard Holes

The size of Hole Wizard holes can be configured:

- Manually, by clicking **Configurations** in the **Hole Specification** PropertyManager and selecting **This configuration**, **All configurations**, or **Specify configurations**.
- In design tables, using parameter syntax **$HW-SIZE@**<*feature_name*>.

Modify Dialog Box

When editing a dimension in the **Modify** dialog box, specify which configurations the change applies to by selecting on the flyout button:

This Configuration

All Configurations

Specify Configurations

If you select **Specify Configurations**, the new **Modify Configurations** dialog box appears, and you can specify different values for each configuration. See **Creating and Modifying Configurations** on page 7-4.

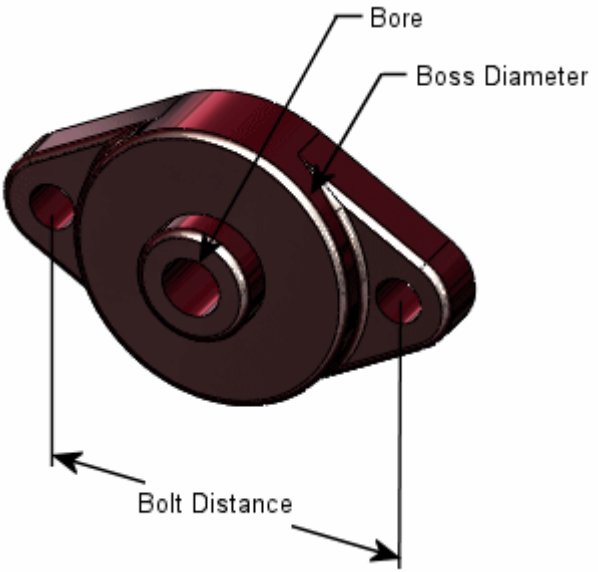
Creating a PropertyManager to Configure Components

For parts that have more than one configuration, you can build a PropertyManager that enables you to select the configuration of a part when you place it in an assembly (similar to the way you select Toolbox parts when you drag them into assemblies).

To create a configuration PropertyManager:

1. Open **Configurations/two_bolt_flange.sldprt**.
2. Save the part as **MyFlange.sldprt**.

The part has seven configurations that vary many dimensions. In this example, you set up a PropertyManager for the following parameters:



3 Right-click the part icon at the top of the FeatureManager design tree and select **Create PropertyManager**.

The **Create PropertyManager** dialog box appears. The part's configured parameters are listed in the left pane. A PropertyManager preview appears in the right pane.

4 In the left pane, for **Bore@Sketch4**:

a) For **Display State**, select **Enabled**.

b) For **Label**, type **Bore**, and press **Enter**.

A selection box labeled **Bore** appears in the PropertyManager preview. The selections in the drop-down list correspond to the values for **Bore@Sketch4** in each configuration of the part.

5 In the left pane, for **L@Sketch1**:

a) For **Display State**, select **Enabled**.

b) For **Label**, type **Boss Diameter**, and press **Enter**.

6 In the left pane, for **J@Sketch1**:

a) For **Display State**, select **Reference**.

b) For **Label**, type **Bolt Distance**, and press **Enter**.

Because you selected **Reference**, the field added to the PropertyManager for **Bolt Distance** cannot be edited.

7 Click **Consolidate**.

The the fields you activated move to the top of the list.

8 Under **Order**, change the value for **L@Sketch1** to **1**, and press **Enter**.

L1@Sketch1 moves to the top of the list in the left pane, and **Boss Diameter** moves to the top in the PropertyManager preview.

9 Click **OK**.

10 Save the part.

See **Create PropertyManager** in the help.

To use the PropertyManager you created:

1 Open a new assembly.

2 In the PropertyManager:

a) Select **MyFlange**.

b) Click ✔.

The **Configure Component** PropertyManager appears with the fields you created.

3 Under **Parameters**, make selections in **Boss Diameter** and **Bore**.

4 Click ✔.

See **Configure Components** in the help.

Creating and Modifying Configurations

The **Modify Configurations** dialog box facilitates creating and modifying configurations for commonly configured properties in parts and assemblies. You can add, delete, and rename configurations and change which configuration is active.

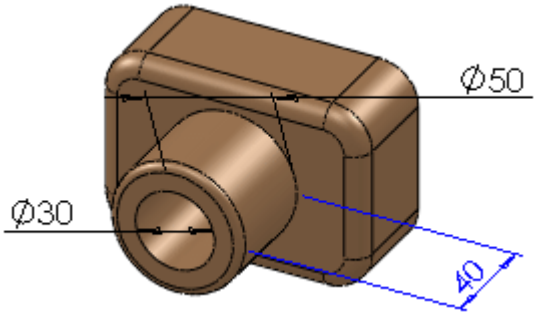
Parts

For features and sketches in parts, you can configure:

- Dimensions
- Suppression states

To configure dimensions:

1. Open **Configurations\block02.sldprt**.
2. In the FeatureManager design tree, right-click **Annotations** and select **Show Feature Dimensions**.



3. In the graphics area, right-click **Ø30** and select **Configure dimension**.

   The **Modify Configurations** dialog box appears. It lists the configurations of the part in one column, and the values of the selected dimension in another column.
4. In the graphics area, double-click **40** and then **Ø50**.

   Columns for the dimensions appear in the dialog box.
5. In the dialog box, click ***<Creates a new configuration.>***.
6. Type **small** and press **Enter**.
7. Repeat step 6 to create a configuration named **large**.
8. In the dialog box, change the dimensions as follows:

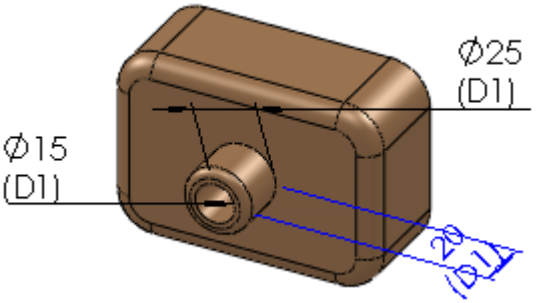
	Sketch3 D1	Extrude2 D1	Sketch2 D1
Default	30	40	50
small	15	20	25
large	35	80	60

9. Right-click **small** and select **Switch to Configuration**.

   The **small** configuration becomes the active configuration.
10. In the lower left corner of the dialog box, click **Rebuild active configuration**.

    The part updates to the new configuration.

11 Click **OK**.



Assemblies

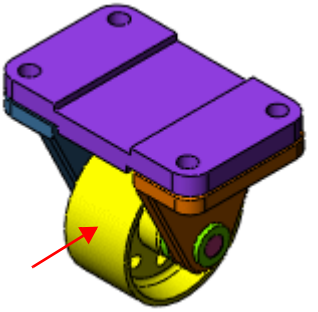
In assemblies, you can specify:

- Which configurations of components to use
- The suppression states of components, assembly features, and mates
- Dimensions of assembly features and mates

To configure components in an assembly:

1 Open **Configurations\castor.sldasm**.

2 In the graphics area, right-click the wheel and select **Configure component**.

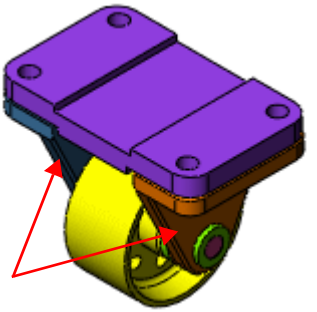


3 In the dialog box, click ***<Creates a new configuration.>***.

4 Type **Medium** and press **Enter**.

5 In the **Configuration** column for **Medium**, click and select **A2**.

6 In the graphics area, double-click each axle support.



Columns are added to the dialog box for the axle supports.

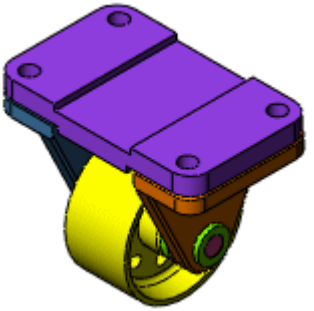
7 For **Medium**, select **D4** in the **Configuration** column for each axle support.

8 Right-click **Medium** and select **Switch to Configuration** to make **Medium** the active configuration.

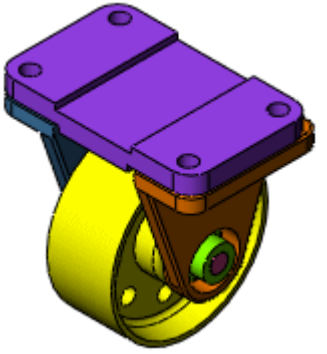
9 Click **Rebuild active configuration** in the dialog box.

The assembly rebuilds with the configurations of the components you selected.

10 Click **OK**.



Default



Medium

See **Modify Configurations** in the help.

8

Motion Studies

This chapter describes enhancements to Motion Studies in the following areas:

- Motion Studies Interface
- MotionManager Enhancements
- Assembly Motion
- Physical Simulation
- COSMOSMotion

Motion Studies Interface

Motion Studies uses the MotionManager, a key frame and timeline based interface (adapted from SolidWorks **Animator)**, and includes:

- **Assembly Motion**. Creates animations of SolidWorks models. Formerly called SolidWorks **Animator** and available as an add-in with SolidWorks Office Premium, it is now available in core SolidWorks.
- **Physical Simulation**. Simulates some of the effects of physics on assemblies.
- **COSMOSMotion**. Simulates and analyzes more complex effects of physics on the motion of an assembly.

See **Introduction to Motion Studies** in the help.

To access Motion Studies:

1 Click the **Motion Study** tab at the bottom left of the graphics area.

If the tab is not visible, click **View**, **MotionManager**.

2 In **Type of Study**, in the top left corner of the MotionManager, select from:

- **Assembly Motion**
- **Physical Simulation**
- **COSMOSMotion**

Levels of Functionality

The functionality is additive based on the study type you select. **Assembly Motion** has the basic level of functionality, **Physical Simulation** includes additional functionality, and **COSMOSMotion** includes all the functionality available in **Motion Studies**.

For performance reasons, choose the lowest level that provides the functionality you need for your study. Higher levels provide for greater levels of realism and give more accurate simulations, but they use more complex calculations, which may take more time.

MotionManager Enhancements

Collapsible Pane

Click **Collapse Motion Study** to show just the toolbar.

Filters

In the MotionManager, click one of the pre-defined filter types to filter the **Motion Studies** FeatureManager design tree.

- **No Filter**. Shows all items in the FeatureManager design tree.
- **Filter Animated**. Shows all items that move or change during the Motion Study.
- **Filter Driving**. Shows all items that cause motion or other changes during the Motion Study.
- **Filter Selected**. Shows only items you select in the FeatureManager design tree (items must be selected first).

Keys Points

At key points in simulations, you can:

- Modify motor parameters.
- Modify the magnitude of a force (**COSMOSMotion** only).
- Suppress and unsuppress mates

Select **AutoKey** to have SolidWorks place a key at the current timebar location for every moved component, as it did in the 2007 release. Clear **AutoKey** to manually place keys in the simulation.

Click **Add/Update Key** to insert or update a key for the selected component at the current timebar location.

Playback Speed

Set the **Playback Speed** you want from the list. This changes only the rate at which the captured frames are played back. It does not change how many frames per second are captured.

Save Animation

You can schedule batch processes to save animations by clicking **Schedule** in the **Save Animation** dialog box. This is useful when saving large animations, which can be a resource-intensive operation. See **SolidWorks Task Scheduler** on page 13-20.

Assembly Motion

Adding Motors to Animations

You can add linear or rotary motors to animations and control them from the **Motor** PropertyManager.

Hands-on Example

Physical Simulation

Contacts

Click **Contacts** to define sets of components to check for contact between. During a motion study, if components within a set contact each other, the contact is detected, and the components react with appropriate motion. If components not grouped together in a set contact each other, the contact is ignored, and they pass through each other.

Springs

With the **Spring** PropertyManager, you can add linear and torsional springs. You can also add damper properties to springs by selecting **Damper** in the PropertyManager and entering values. Torsional springs and all dampers can be used only in **COSMOSMotion**.

COSMOSMotion

To activate **COSMOSMotion**, select it in the **Add-Ins** dialog box.

Analytical Properties of Mates

Load References have been replaced by **Load Bearing Faces**. You define **Load Bearing Faces**, **Friction**, and **Bushings** on the **Analysis** tab of the **Mate** PropertyManager. See **Analysis Tab** on page 6-11.

Dampers

In addition to adding dampers directly to springs, you can also click **Damper** to create stand-alone dampers with the **Damper** PropertyManager.

Fixed and Floating Parts

The terms **Ground Parts** and **Moving Parts** have been replaced with **Fixed** and **Floating**, respectively, and are automatically detected based on the mates in your assembly.

Legacy Studies

Assemblies containing a **Simulation** made with **Physical Simulation** in a previous release of SolidWorks open with a **Physical Simulation** tab on the MotionManager.

With **COSMOSMotion** added in, assemblies containing a **Simulation** made with **COSMOSMotion** in a previous release of SolidWorks open with a **COSMOSMotion** tab on the MotionManager, and the **COSMOSMotion IntelliMotion Browser** appears in place of the FeatureManager design tree in the MotionManager. You cannot change the simulations, but you can view and run them.

Plots

When you plot results, a triad appears on the component or mate to show its local direction of X, Y, and Z.

You can hide a plot by clicking in its upper right corner. To show it again, expand the **Results** folder in the FeatureManager design tree, right-click **XY Plots** , and select **Show**.

Redundant Constraint Handling

You have a choice of two options to handle redundant constraints.

To specify redundant constraint handling:

1. Click **Properties** (MotionManager toolbar).
2. Under **COSMOSMotion**, click **Advanced Options**.
3. In the **Advanced Simulations Options** dialog box, select one:
   - **Automatically delete redundant constraints**. Deletes all redundant constraints, leaving a fully constrained assembly. The remaining constraints support the loads in the system.
   - **Replace redundant constraints with bushings**. Replaces with a bushing, any mate that has a redundant component. The loads in the system are distributed based on which mates are replaced.

See **Redundant Constraints** in the help.

9

Drawings and Detailing

This chapter describes enhancements to detailing in the following areas:

- General
- Drawing Sheets
- Drawing Views
- Bills of Materials
- Tables

General

Balloons in Notes

You can now add balloons to notes. When creating or editing a note, click a balloon to insert it. Any modification to the balloon's properties from the **Balloon** PropertyManager updates the balloon in the note. The font size of an individual balloon can be adjusted after it has been inserted as though it were any other font character.

Note referring to balloon item number 2.3, with corresponding pentagon-shaped balloon inserted in note text.

Balloon Text

Balloon text links parametrically to the BOM. Changing the item in the BOM propagates to the balloon.

Dialog Box Removal

The **Dimension** PropertyManager now contains the functionality that was in the **Dimension Properties** dialog box. The dialog box and the **More Properties** button from the **Dimension** PropertyManager have been removed.

The **Note** PropertyManager now contains the functionality that was in the **Properties** dialog box for notes. The dialog box has been removed.

Dimension Alignment

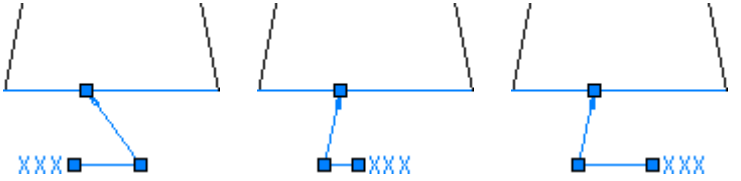
You can now align dimensions to edges. Place the dimension, right-click it, select **Align to edge**, then select an edge in the drawing.

Dimension Properties

You can now undo modifications to dimension properties.

Leaders

You can now use drag handles to change bent leader lengths for annotations.



Drawing Sheets

New Drawings from Open Documents

The **Make Drawing from Part/Assembly** tool is available on the flyout menu for **New** on the Menu Bar.

Copying Sheets

You can copy entire drawing sheets inside of a drawing document or across different drawings. Use:

Supported Copy and Paste Methods	Supported To and From Locations
Edit, **Copy**/**Paste**	FeatureManager design tree
Right-click menus	Graphics area of a sheet
Ctrl + drag (between tiled drawing windows)	Sheet tabs

See **Copying Sheets** in the help.

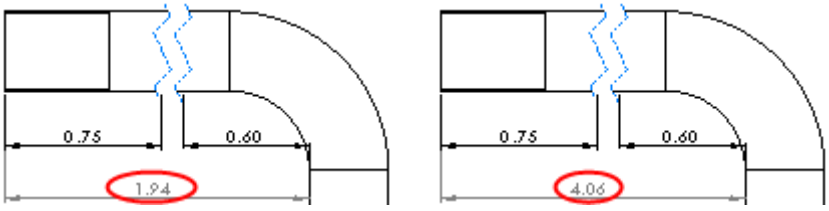
Inserting Images

You can insert pictures into a drawing just as you would in a sketch. Click **Insert**, **Picture**, and browse to your picture. Open the picture and use the **Sketch Picture** PropertyManager to adjust the picture.

Drawing Views

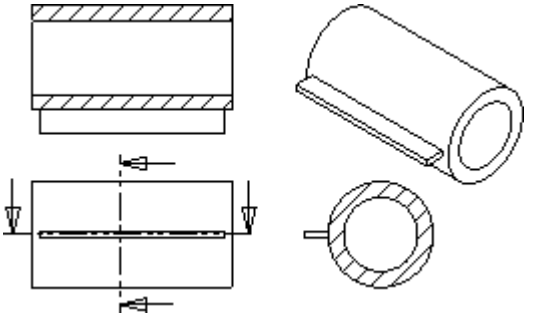
Broken Views

You can lock break lines in place. After breaking the view, dimension the break lines to a known portion of the geometry. If the overall dimension changes, the break lines stay in place. The dimensions for the break lines are hidden if the break lines are not active. These dimensions are only for use in the drawing document and do not appear on a printed drawing.



Section Views

You can exclude ribs from sectioning in drawings. From the **Section View** dialog box, select a rib to add it to **Excluded components**.

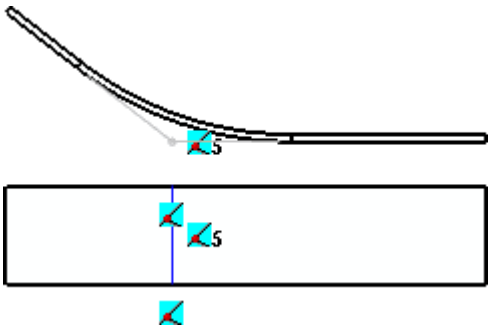


Annotation Views

Views with annotations are marked with an icon in the **Model View** PropertyManager's **Orientation** section; select **Import annotations** to bring the annotations into the drawing.

Sketch Entity Alignment

You can constrain sketch entities to geometry in multiple drawing views. Sketch objects in the drawing and use constraints as you normally would in a sketch.



This image shows:

- A sketched point, in the top view, constrained to the theoretical sharp of the part using intersection relations (relations not shown for clarity).
- A sketched line, in the bottom view, constrained to be coincident with the bottom view's geometry and the sketch point in the top view.

Bills of Materials

Column Contents

Double-click a column header to display a list of choices for the column contents. Custom properties defined in at least one part in the assembly, are available in the list.

Property Modifications

Changes to cells containing **Custom Properties** or **Part Numbers** now update in the model's **Properties**.

See **Bill of Materials Overview** in the help.

For any part which does not yet have the Custom Property defined, an edit to the cell adds a definition for this property to the part's properties, on the **Configuration Specific** tab.

To view the **Configuration Specific** tab, click **File**, **Properties**.

When you update the BOM for a part that already has the Custom Property defined, the link between the part's properties and the Bill of Materials (BOM) updates.

If you edit the **Part Number**, the new value appears as a **User Specified Name** in the part's **Configuration Properties** PropertyManager.

To view the part's **Configuration Properties** PropertyManager, click the part's **Configuration** tab, right-click the configuration you are using, and select **Properties**.

Virtual Components

When the **BOM Type** is set to **Parts only** or **Indented assemblies**, virtual components display this symbol on the BOM. (See **Virtual Components** on page 6-7).

Weldment Material Usage

In a BOM, right-click anywhere in a row containing a weldment and select **Dissolve Weldment** to show the material usage of the weldment. The weldment's cut list is rewritten for material usage, such that, each different type of material in the cut list and the total amount of that material type is shown (for example, total length of square tubing used in all the weldment's components). Collapse the weldment by right-clicking on any of its component and selecting **Restore Weldment**.

You can do this for all the weldments in a BOM at the same time by activating the BOM, then right-clicking and selecting **Dissolve Weldment**.

Tables

Editing Cells

Double-clicking a cell brings up the context toolbar with items specific to cell editing.

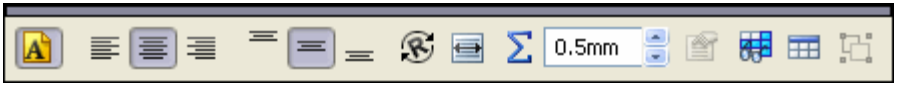
Editing Cells Externally

Items linked to properties in tables (for example, revision level) are directly editable from Windows Explorer. See **Property Modifications** on page 9-5, and **Modifying Document Properties** on page 1-10.

Editing Tables

Hovering over a column divider changes the pointer, allowing you to drag or **Alt** + drag to resize the column to the left or right, respectively, of the divider. Double-clicking a row or column divider automatically resizes the row or column.

Hovering over a cell changes the pointer to . Click to display the context toolbar. The toolbar's buttons reflect the options available for the type of table and selections (rows, columns, and cells).

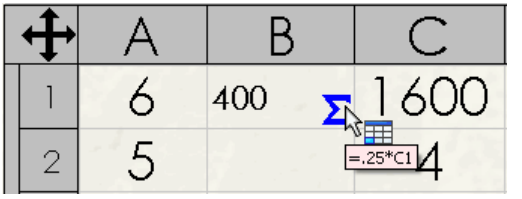


Use the context toolbar to make formatting and font changes without having to use PropertyManagers. The **Row**, **Column**, and **Cell** PropertyManagers have been eliminated, and their functionality put into the context toolbar.

To access a table's PropertyManager, click the move table icon in the upper left corner or right-click the table and select **Properties**.

Equations in Cells

When a table is active, cells that contain equations display the equation symbol. Hover over the symbol to display the equation. Click the symbol to open the equation editor.



Fit Text to a Cell or Note

You can compress text to fit in a table cell or note.

To compress text to fit the cell size

1 Select the table cell or note to modify.

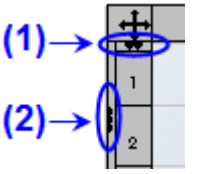
2 Click **Fit Text** (context toolbar).

The cell or note automatically resizes, and the font portion of the context toolbar appears.

To turn off the fit text feature in notes or table cells, click **Fit Text** .

Header and Balloon Status

With a table active, click the expansion arrows (1) to show the table header. For a BOM, click the expansion arrows (2) to show the balloon status and assembly structure.



Header Position

To toggle the table header, click **Table Header Top** or **Table Header Bottom** (context toolbar).

Hide and Show Rows and Columns in Tables

You can hide and show rows and columns in tables with the context toolbar. Click **Hide/Show** and select the headers of the rows or columns you want hidden. The selected rows and columns are highlighted to show they will be hidden when you click **Hide/Show** again.

Making Font Changes

Click to clear **Use document font** (context toolbar) to display the font portion of the context toolbar.

Modifying Columns

You can drag columns by their headers to reorder them.

A toolbar that lists the options for the column's contents is displayed by double-clicking a BOM's column header or selecting the column and clicking **Column Property** (context toolbar).

10

Dimensioning and Tolerancing

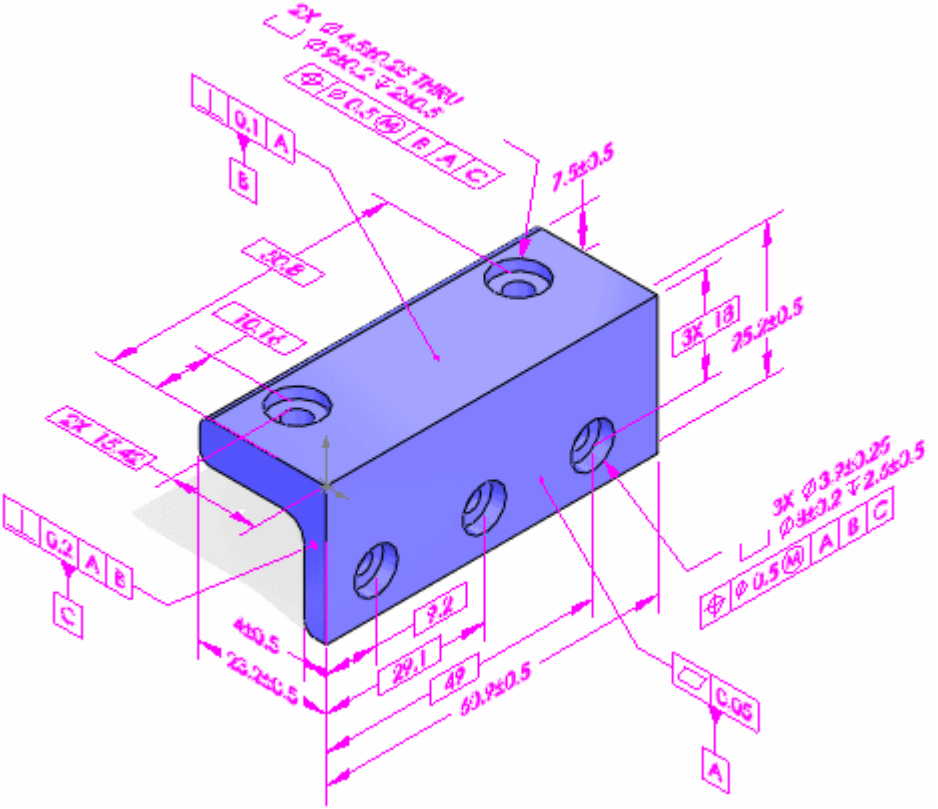
This chapter describes enhancements to tolerancing in the following areas:

- Overview
- DimXpert for Parts
- TolAnalyst™

Overview

Geometric Dimensioning and Tolerancing (GD&T) offers many benefits:

- The design language is standardized.
- Design intent is clear and precise to the customers, suppliers, and production teams.
- You can calculate the worst-case mating limits.
- Through the use of datums, the production and inspection processes are repeatable.
- Assembly is assured from qualified production parts.



Two new GD&T-based applications are available:

- **DimXpert** - Places dimensions and tolerances on parts.
- **TolAnalyst** - Performs a stack-up analysis on assemblies to assure the components can be assembled.

DimXpert for Parts

DimXpert for parts is a set of tools that applies dimensions and tolerances to parts according to the requirements of the ASME Y14.41-2003 standard.

Set **Dimensioning standard** to **ANSI** under **Tools**, **Options**, **Document Properties**, **Detailing** to display annotations according to the ASME standard. Standards other than ANSI are not yet supported.

Features

For DimXpert, features means manufacturing features. For example, in the CAD world, you create a "shell" feature, which is a type of "pocket" feature in the manufacturing world.

Supported manufacturing features:

- Boss
- Chamfer
- Cone
- Cylinder
- Discrete feature types
- Fillet
- Counterbore hole
- Countersink hole
- Simple hole
- Intersect line
- Intersect plane
- Intersect point
- Notch
- Plane
- Pocket
- Slot
- Surface
- Width

When you apply DimXpert dimensions to manufacturing features, DimXpert uses these methods in this order to recognize features:

1. Model feature recognition
2. Topology recognition

Model Feature Recognition

The benefit of model feature recognition is that identified features update if you modify the model feature, especially if you add features or faces. DimXpert recognizes these design features:

- Chamfer
- Cosmetic thread
- Certain extrudes (for pattern extraction)
- Fillet
- Hole Wizard hole
- Simple hole
- Certain patterns (linear, circular, and mirror for pattern extraction)

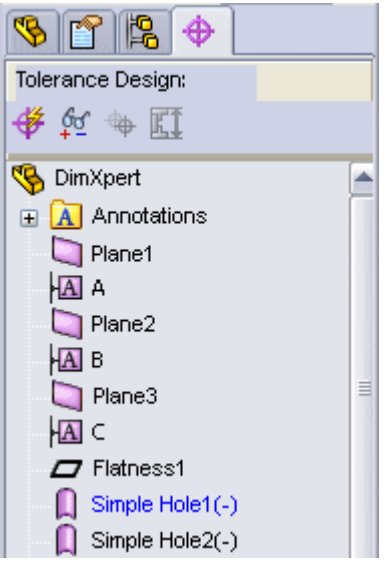
Topology Recognition

If model recognition fails to recognize features, DimXpert uses topology recognition. The benefit of topology recognition is that it recognizes manufacturing features that model recognition cannot, such as slots, notches, and pockets. Only topology recognition is used for features on imported bodies. Topology features update if you make geometry changes, but new instances are not added to pattern features.

Using DimXpert

A suite of DimXpert tools lets you insert dimensions and tolerances manually or automatically. The DimXpertManager:

- Lists the tolerance features defined by the DimXpert in chronological order.
- Displays DimXpert tools.



To set the DimXpert options:

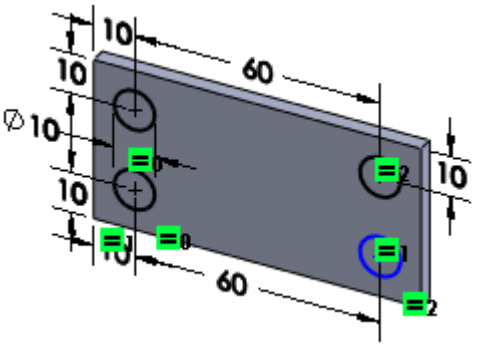
1. Open **DimXpert.sldprt**.
2. Click **Tools**, **Options**, **Document Properties**. Under **DimXpert**, select **Geometric Tolerance**.
3. Under **Basic dimensions**, select **Create basic dimensions**.
4. Click **OK**.

   You can set other DimXpert tool options under **DimXpert**.

To inspect the model:

1 In the FeatureManager design tree, expand **Cut-Extrude1** and edit **Sketch2**.

Note the extruded cut dimensions. The part is 80mm X 40mm.

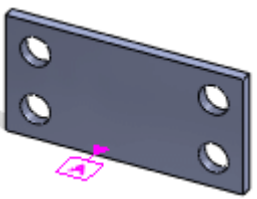


2 Exit the sketch.

To manually insert dimensions and geometric tolerances using DimXpert:

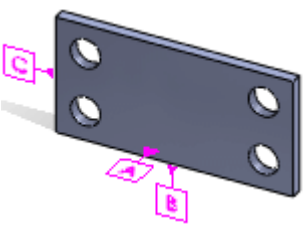
1 Click **Datum** (DimXpert toolbar) or **Tools**, **DimXpert, Datum**.

2 Click the face shown to place datum **A**, then drag and position it.



3 Add datums **B** and **C** as shown, then click ✓.

The DimXpertManager updates to show the datums.



> If a tolerance is not properly oriented, reorient it. Right-click the annotation, select **Change Annotation View**, **By Selection**. Select the proper face, which must be perpendicular to the datum plane, then click ✓.

4 Click **Size Dimension** (DimXpert toolbar) or **Tools**, **DimXpert**, **Size Dimension**.

> DimXpert dimensions are driven. They cannot be driving dimensions.

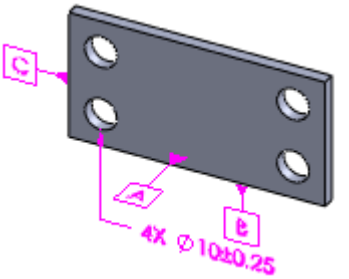
5 Select the lower-left hole.

In the context toolbar, **Pattern** is automatically selected through model feature recognition because the four same-sized holes reside on a single cut.

6 Click in the graphics area to place the size dimension.

7 Click ✓.

DimXpert recognized the four holes as a **Hole Pattern** feature in the DimXpertManager.



The icon under **Hole Pattern** indicates DimXpert used model feature recognition. If topology recognition is used, no icon appears.

8 Click **Geometric Tolerance** (DimXpert toolbar) or **Tools**, **DimXpert, Geometric Tolerance**.

9 In the dialog box:

a) Select **Position** in **Symbol**.

b) Type **0.5** for **Tolerance 1**.

c) Click **Diameter** ∅.

The pointer must be in the **Tolerance 1** box.

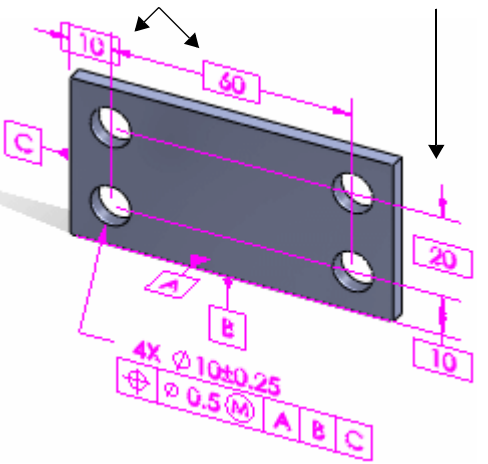
d) Click **Maximum Material Condition** Ⓜ.

e) Type **A**, **B**, and **C** for **Primary**, **Secondary**, and **Tertiary**.

f) Select the lower-left hole with the hole pattern, then click anywhere in the graphics area to place the geometric tolerance with the pattern dimension.

g) Click **OK**.

DimXpert creates basic dimensions for the hole location along the top and right faces.



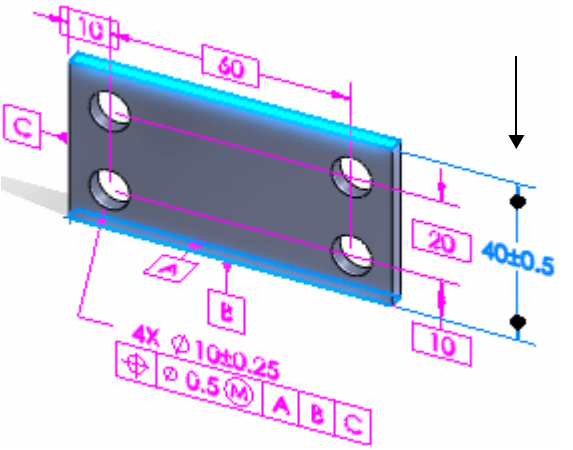
Basic dimensions are boxed.

For example 60.

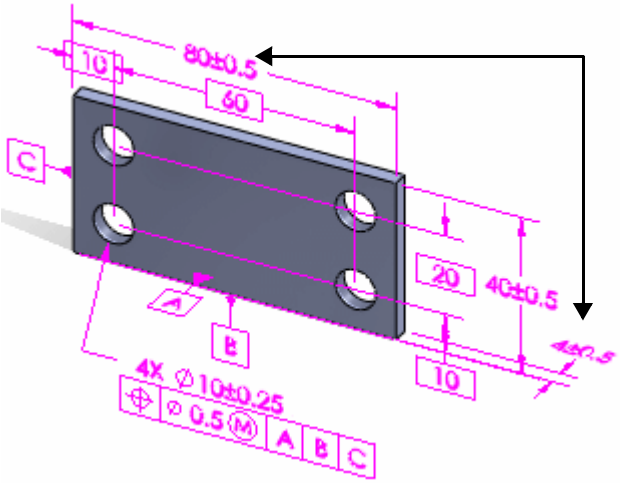
If you do not see the basic dimensions, make sure you have set the DimXpert options correctly. See **To set the DimXpert options:** on page 10-4.

10 Click **Location Dimension** (DimXpert toolbar) or **Tools**, **DimXpert**, **Location Dimension**.

11 Select the top and bottom faces, then click to position the dimension.



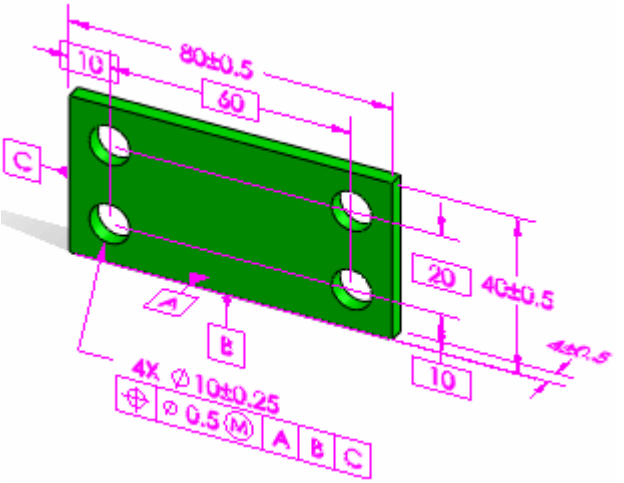
12 Use **Location Dimension** to dimension the right and left faces, then the front and back faces. Click .



The model should resemble this image.

13 Click **Show Tolerance Status** (DimXpert toolbar) or **Tools**, **DimXpert**, **Show Tolerance Status** to see which features are under or over constrained for size and location.

- Under constrained features are yellow. The features are marked with a (-) sign in the DimXpertManager.
- Fully constrained features are green.
- Over constrained features are red and marked with a (+) sign.



The model is fully constrained.

14 Click **Delete All Tolerances** (DimXpert toolbar) or **Tools**, **DimXpert**, **Delete All Tolerances** to delete all DimXpert tolerances and dimensions from the model and from the DimXpertManager. You can also right-click and delete individual items in the DimXpertManager.

15 Click **Tools**, **Options**, **Document Properties**. Under **DimXpert**, select **Display Options**.

16 Under **Redundant dimensions**, select **Show instance count**.

17 Click **OK** to set up the part for the next procedure.

18 Keep the part open for the next procedure.

To automatically insert dimensions and tolerances using DimXpert:

1 Click **Auto Dimension Scheme** (DimXpert toolbar) or **Tools**, **DimXpert**, **Auto Dimensions Scheme**.

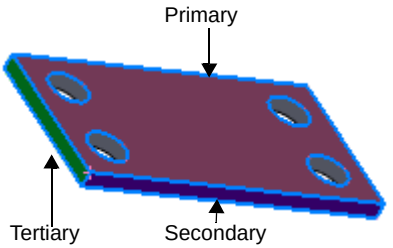
2 In the PropertyManager:

a) Under **Settings** select:

- **Prismatic** for **Part type**.
- **Plus and Minus** for **Tolerance type**. The plus-minus tolerance scheme uses linear dimensions to locate all feature types.

b) Under **Reference Features** select:

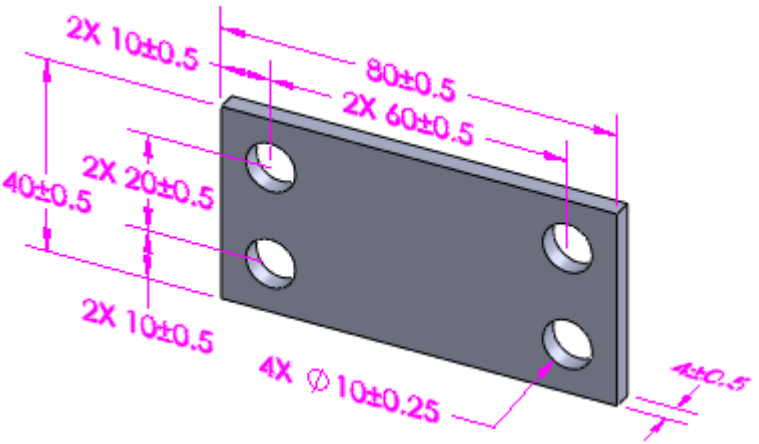
- The front face for **Primary**.
- The bottom face for **Secondary**.
- The left face for **Tertiary**.



The faces use the **Reference Features** colors from the PropertyManager.

c) Make sure to select **All features** under **Scope** and all items under **Feature Filters**.

d) Click .



DimXpert automatically inserts dimensions and tolerances, using plus-minus terminology, to fully constrain the model.

The dimensions and tolerances shown may vary from your display.

3 Delete all DimXpert dimensions and tolerances.

Now re-run the DimXpert to use a geometric tolerance scheme, which uses position, circular runout, and surface profiles as applicable.

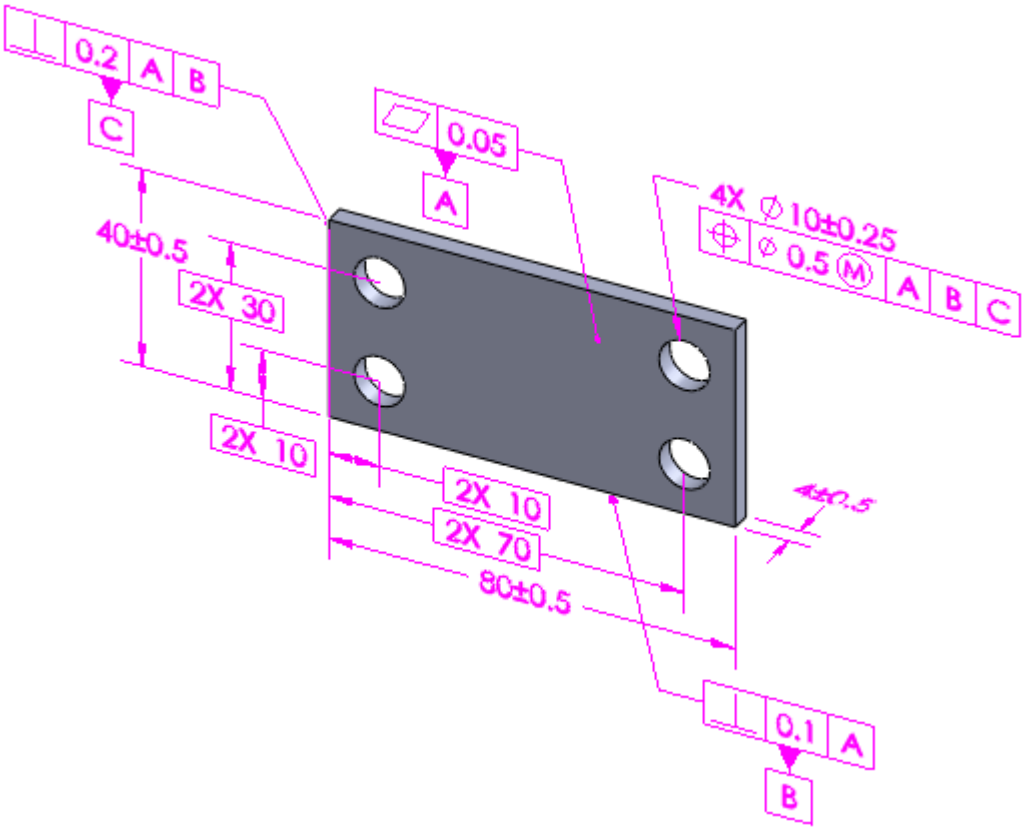
Auto Dimension Scheme applies tolerances of form (flatness only), orientation, and location as required to interrelate the datum features.

4 Click **Auto Dimension Scheme** .

5 Under **Settings**, select **Geometric** for **Tolerance type**.

6 Select the same faces used previously for **Reference Features**.

7 Click .



For legibility, the images were captured with the **Always display text at same size** option selected. Per ASME standards however, annotations should scale in-kind with the geometry. You should therefore clear this option and set an appropriate **Text scale** in the **Annotation Properties** dialog box.

TolAnalyst

TolAnalyst is a tolerance analysis application that determines the effects that dimensions and tolerances have on parts and assemblies. The TolAnalyst tools let you perform “worst-case” tolerance stack-up analysis on assemblies.

First you use the DimXpert tools to apply dimensions and tolerances to the parts or components in an assembly, then you use the TolAnalyst tools to leverage that data for stack-up analysis.

TolAnalyst is available only in SolidWorks Office Premium.

See **TolAnalyst** on page 14-17.

11

COSMOSWorks

This chapter describes enhancements to COSMOSWorks® in the following areas:

- General
- New Study Types
- Analysis Studies
- Loads and Restraints
- Mesh
- Bonding and Contact for Static Studies
- Result Viewing

The following enhancements are available in COSMOSWorks Designer and above unless otherwise noted. Enhancements marked with a (P) are available in COSMOSWorks Professional and above. Enhancements marked with an (A) are only available in COSMOSWorks Advanced Professional.

General

- A 64-bit version of COSMOSWorks is now available.
- When you select a face in the graphics area, the **Beams**, **Shells**, or **Solids** folder in the COSMOS® AnalysisManager expands and highlights the associated body or component.
- **N/mm^2 (MPa)** units are now available in many places, including: **Pressure** PropertyManager, **Stress Plot** PropertyManager, **Options** dialog box, and the three tabs of the **Material** dialog box.
- A new CommandManager helps you access the COSMOSWorks interface easily. The CommandManager is context-sensitive to the active study's type and status.
- (A) You can use the nitinol material model for shells in nonlinear studies.
- New toolbars are added for **COSMOSWorks Dynamics** and **COSMOSWorks Trend Tracking**.
- No grayed out commands are shown in shortcut menus.
- You can search for analysis keywords within analysis files, including *.**sldalasm**, *.**sldalprt**, *.**sldasm**, and *.**sldprt** files.

See **Searching Analysis Documents** in the help.

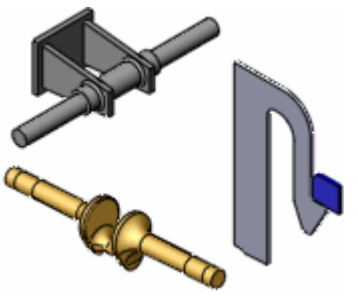
- New NAFEMS linear static benchmarks are added to the verification list.
- The **Report** tool has been redesigned for COSMOSWorks 2008.

See **Study Reports** in the help.

Analysis Advisor

The Analysis Advisor in the Task Pane has been enhanced to include:

- Information to assist you with loads, restraints, and contact conditions.
- A database of load, restraint, and contact condition examples. Some examples are shown. You can customize the database of examples to add, edit, or remove examples.

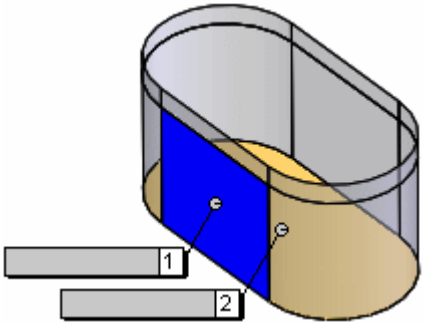


Hyperelastic Mooney-Rivlin and Ogden Material (A)

You can view the constants calculated by the program when using a stress-strain curve for Mooney-Rivlin or Ogden material models for nonlinear studies. The constants are saved in a text file with the extension *.log* in the active result's folder for the study.

Shells Displayed by Thickness or Material

You can display shells by color to identify the thickness or material of the shell. Right-click the **Shells** folder in the COSMOS AnalysisManager and select the appropriate option. In the image, one wall is 1" thick, the bottom is 2" thick, and the remaining walls are unassigned and therefore transparent.



You can also create a mesh quality plot showing the thickness or material of the shells. To create this plot, right-click **Mesh** and select **Create Mesh Plot**.

New Study Types

Pressure Vessel Study (P)

In a new Pressure Vessel study, you combine the results of two or more static studies to determine the worst case condition. Each static study has a different set of loads that produce corresponding results. The Pressure Vessel study combines the results of the static studies algebraically.

See **Pressure Vessel Studies** in the help.

Linear Dynamic Study (A)

Static studies assume that loads are constant or applied very slowly until they reach their full values. Because of this assumption, the velocity and acceleration of each particle of the model is assumed to be zero. As a result, static studies neglect inertial and damping forces.

For many practical cases, loads are not applied slowly or they change with time or frequency. For such cases, use a dynamic study.

Linear dynamic studies are based on frequency studies. The software calculates the response of the model by accumulating the contribution of each mode to the loading environment. In most cases, only the lower modes contribute significantly to the response. The contribution of a mode depends on the load's frequency content, magnitude, direction, duration, and location.

The frequency of the highest mode included in the analysis may have to be higher than the frequencies of the loads. The mass participation factors calculated in frequency studies can help you decide how many modes to include in the analysis. As a general guideline, some codes recommend a minimum of 80% mass participation in the direction of motion.

With the guidelines above, running several iterations by increasing the number of modes helps you in assessing convergence to the correct results.

Dynamic loads

Dynamic loads can be generally classified as deterministic or non-deterministic. Deterministic loads are well-defined as functions of time and can be predicted precisely. They can be harmonic, periodic, or non-periodic. If the loads are deterministic, the results are also deterministic. Non-deterministic loads cannot be well-defined as explicit functions of time, and they are best described by statistical parameters. If the loads are non-deterministic, the results are also non-deterministic.

In dynamic studies, the energy of vibrating systems is dissipated by various damping mechanisms. The following types of damping are available:

- Modal
- Rayleigh
- Composite modal
- Concentrated dampers (for modal time history studies only)

See **Damping Effects** in the help.

To handle different loading environments, COSMOSWorks offers 3 types of linear dynamic studies:

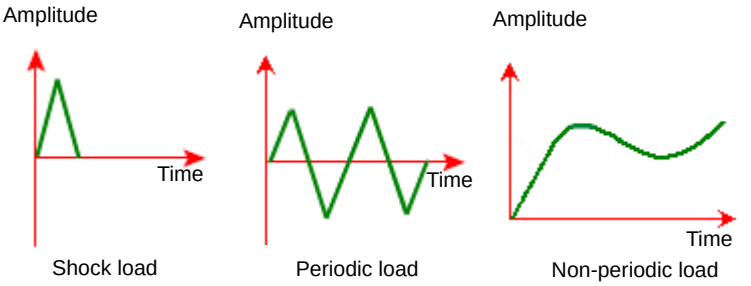
- Modal Time History
- Harmonic
- Random Vibration

Modal Time History Analysis

Use modal time history analysis when the variation of each load with time is known explicitly, and you are interested in the response as a function of time. Typical loads include:

- shock (or pulse) loads
- general time-varying loads (periodic or non-periodic)

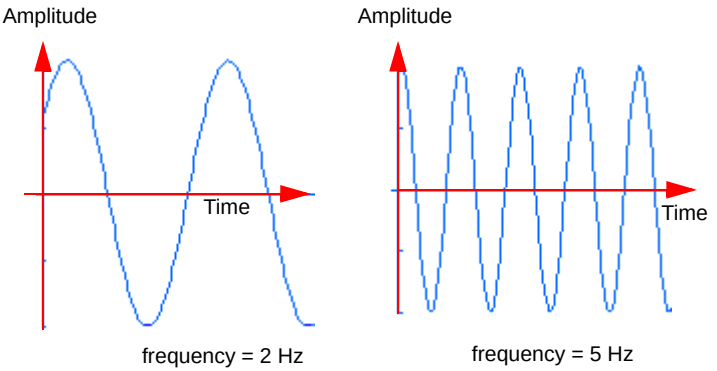
- uniform base motion (displacement, velocity, or acceleration applied to all supports)
- support motions (displacement, velocity, or acceleration applied to selected supports)



After running the study, you can view displacements, stresses, strains, reaction forces, etc. at different time steps, or you can graph results at specified locations.

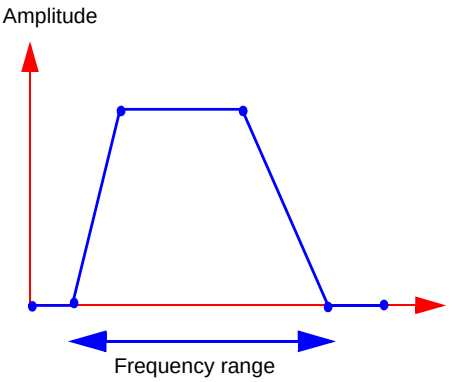
Harmonic Analysis

Use harmonic analysis when your model is subjected to harmonic loads only. A harmonic load P is expressed as $P = A \sin(\omega t + \Phi)$ where: A is the amplitude, ω is the frequency, t is time, and Φ is the phase angle. Sample harmonic loads versus time with same magnitude A, different frequencies and phase angles are shown below:



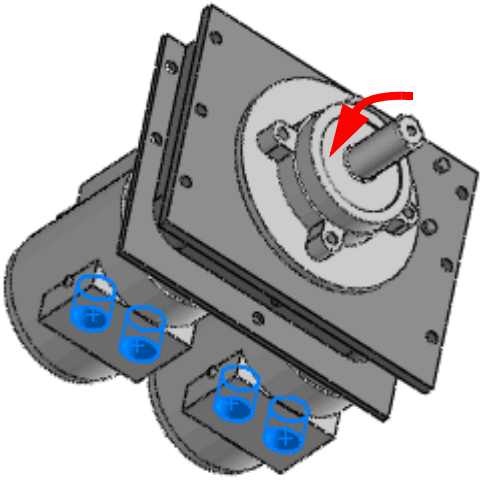
Although you can create a modal time history study and define loads as functions of time, you may not be interested in the transient variation of the response with time. In such cases, you save time and resources by solving for the steady-state peak response at the desired operational frequency range using harmonic analysis.

A sample input for a harmonic analysis is shown in the figure. The peak amplitudes of harmonic loads are plotted versus the operating frequencies.



After running the study, you can view peak stresses, displacements, accelerations, and velocities over the range of operating frequencies.

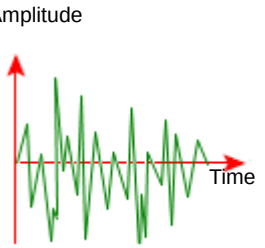
For example, a motor mounted on a test table transfers harmonic loads to the support system through the bolts. You can model the supporting system and define a harmonic study to evaluate the steady-state peak displacements, stresses, etc. for the motor's range of operating frequencies.



Random Vibration

Use a random vibration study to calculate the response due to non-deterministic loads. Examples of non-deterministic loads include:

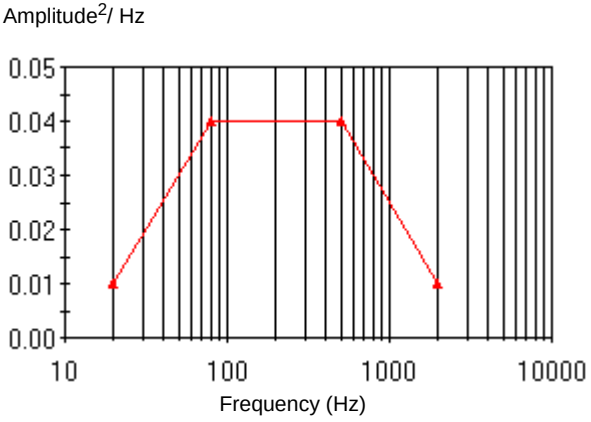
- loads generated on a wheel of a car traveling on a rough road
- base accelerations generated by earthquakes
- pressure generated by air turbulence
- pressure from sea waves or strong wind



A sample random load versus time is shown in the figure.

In a random vibration study, loads are described statistically by Power Spectral Density (PSD) functions. The units of PSD are the units of the load squared over frequency as a function of frequency. For example, the units of a PSD curve for pressure are $(psi)^2/Hz$ over Hz.

A sample PSD curve is shown below. The X axis is plotted on a logarithmic scale for a clear illustration of the wide frequency range.



After running the study, you can plot Root-Mean-Square (RMS) or PSD results of stresses, displacements, velocities, etc. at a specific frequency or graph results at specific locations versus frequency values.

See **When to Use Dynamic Analysis** in the help.

Nonlinear Dynamic Study (A)

You cannot use linear dynamic studies if your model has nonlinear materials or undergoes large deformations. Nonlinear dynamic studies are solved in the time domain. Unlike linear dynamic studies, nonlinear dynamic studies do not require the extraction of frequencies and mode shapes. Loads are defined with respect to time similar to modal time history studies.

Note that while time is a pseudo variable used to step loading for most nonlinear static studies, it is always a real variable for dynamic studies.

For example, if your model is assembled with thin-walled parts prone to large deformations, has a hyperelastic material model, and is subjected to an impulse load, a nonlinear dynamic study is recommended.

Example of Linear Dynamic Study (A)

You create a linear dynamic modal time history study to examine the response of a basketball rim subjected to an impulse load caused by a slam dunk.

To simulate the response of a basketball rim under an impulse load:

1 Open **COSMOSWorks\Basketball_rim.sldprt.**

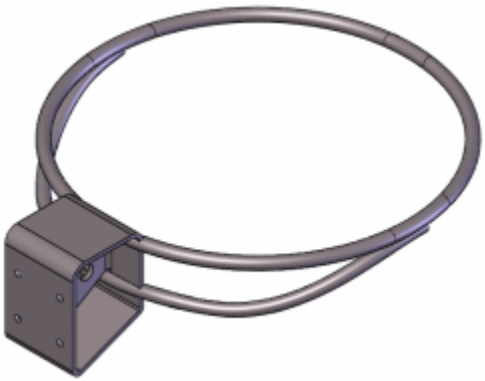
2 To create a linear dynamic study, right-click **Frequency** in the COSMOS AnalysisManager, and select **Copy to New Dynamic Study**.

3 In the dialog box,

a) Type **Dynamic_First** for **Study Name**.

b) Select **Modal Time History Analysis** in **Linear Dynamic Study Type**.

c) Click **OK**.



When you create a dynamic study from an existing frequency study, the material properties, restraints and results of the frequency study are copied to the new dynamic study.

4 To set the properties of the dynamic study, right-click **Dynamic_First** and select **Properties**.

a) In **Frequency Options,** set **Number of frequencies** to **5**.

b) In **Dynamic Options,** set **End time** to **0.8**, and **Time increment** to **0.0005.**

For the solution time step, use a value that is a fraction of the period of the load's highest frequency. It is recommended to run the analysis with decreasing time steps and note the convergence of the results.

5 Right-click **Load/Restraint** and select **Force**.

a) Select the front middle part of the rim for **Faces, Edges, Vertices for Force**.

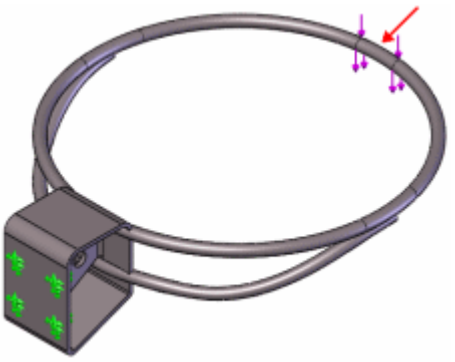
b) Select **Front Plane** in **Face, Edge, Plane, Axis for Direction** box.

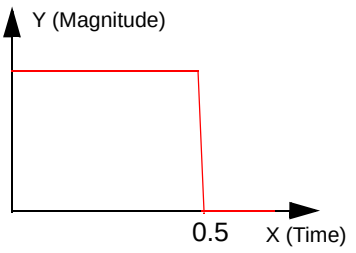
c) Select **Along Plane Dir 2,** and type **260** (lb).

d) Check **Reverse direction**.

e) Under **Variation with Time**, select **Curve** and click **Edit**. Type the following values for the force curve data.

Points	X	Y
1	0	1.0
2	0.4995	1.0
3	0.5	0
4	10	0





f) Click **OK**

g) Click ✓.

6 Right-click **Damping**, and select **Edit/Define**.

a) Under **Options**, select **Modal damping**.

a) Under **Damping Ratios**, type **0.05** in the **Damping Ratio** column.

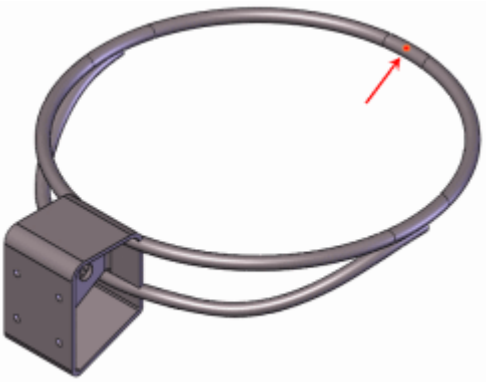
b) Click ✓.

7 Right-click **Result Options**, and select **Edit/Define**.

a) In the PropertyManager under **Save Results**, select **For specified solution steps**.

b) In the **Solution Steps-Set 1** boxes set **Start** to **1**, **End** to **1600,** and **Increment** to **10**.

c) In the **Response Plots** box, select **Point 1**.



d) Click .

If you do not specify any vertex or point in the **Response Plots** box, you can graph the response at any node for the steps requested in the **Result Options** PropertyManager.

8 Mesh the model using a **Global Size** of **0.8** in, and run the analysis.

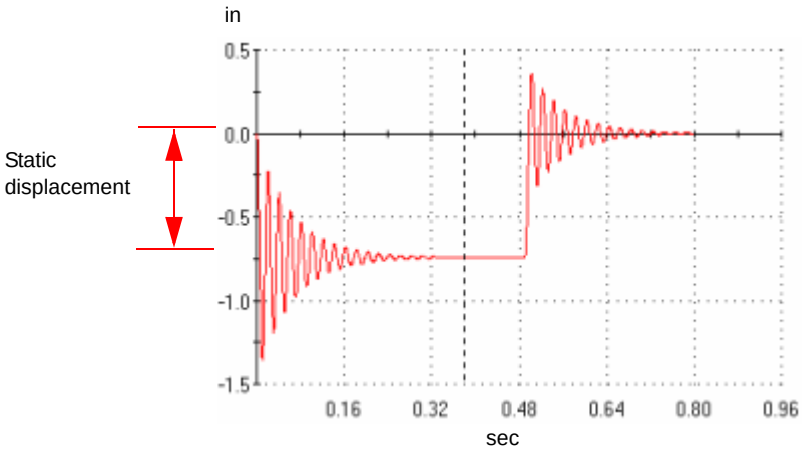
9 To plot a displacement time history at **Point 1**, right-click **Results** and select **Define Response Graph** .

The **Time History Graph** PropertyManager appears with **Point 1** highlighted in the **Predefined locations** box.

a) In the PropertyManager under **Y-axis,** select **Displacement, UY: Y displacement** in **Component** **,** and **in** in **Units** .

b) Click .

The following graph is displayed where X-axis is *sec* and Y-axis is *in*.



- The response graph clearly depicts oscillations of the basketball rim occurring during the slam dunk. The maximum UY displacement is approximately 1.35 in. This value is greater than 0.78 in which is the maximum static UY displacement of the rim (see the UY displacement plot of **Static** study). The hoop comes to rest at the end at about 0.32 sec. After releasing the load, the rim undergoes free vibrations until it recovers its initial position at about 0.8 sec.
- To test the accuracy of the results, run a second dynamic study using 10 modes. The results show minor changes in the response indicating that using 5 modes gives accurate results in this case. In many cases, a larger number of modes may be required.

Analysis Studies

- (P) Calculations of view factors for thermal studies are improved for shells.
- (P) In earlier releases, a face in a thermal study can only radiate to the ambient or to other model faces. In this release, a face can radiate to both the ambient and other model faces simultaneously.
- (A) Large strain formulation for nonlinear studies is enhanced for all material models to improve accuracy and convergence.
- (P) Exporting studies to NASTRAN is improved. The translator now exports the following additional features:
  - mixed mesh studies
  - local restraints
  - node-to-surface and surface-to-surface bonding
  - node-to-node, node-to-surface, and surface-to-surface contact

- remote forces, masses, and displacements
- elastic supports and link connectors

- (A) You can now export static, frequency, buckling, nonlinear, and thermal studies to ABAQUS. In addition to exporting mesh, material properties, loads and restraints, the translator exports the following features:
  - node-to-surface bonding
  - surface-to-surface bonding (introduced in COSMOSWorks 2008)
  - node-to-surface and surface-to-surface contact
  - remote forces, masses, and displacements
  - rigid, elastic support, and pin connectors

To export a study to ABAQUS:

1. Right-click the study's name and select **Export**.
2. In **Save as type**, select **ABAQUS Files(*.inp)**.
3. In **File name**, type a name.
4. Click **Save**.

Beams

- You can mix beams, shells, and solids in a mixed mesh study.
- The software recognizes geometry other than linear structural members as beams. Supported geometry includes: curved structural members, extruded and swept bodies, mirrored and patterned structural members and bodies, and imported geometry. Note that the software cannot recognize all merged and combined geometry as beams.
- You can display the neutral axes of beams. Select **Show beams as lines** under **Results** in the **Edit Joints** PropertyManager.
- You can display the directions used to display beam results for each beam. Select **Show beam orientation** under **Deformed Plot Options** in the **Settings** PropertyManager for any plot.
- You can now apply forces across the full length of a beam or at reference points. The reference points must lie within the model boundary. In earlier releases, you could only apply forces at joints.
- For forces applied across the full length of a beam, you can specify a value of force or force per unit length. Select **Per unit length** under **Units** in the **Force** PropertyManager to specify force per unit length of the beam.
- You can list beam forces and stresses. Right-click the **Results** folder and select **List Beam Forces**. Listed items include: axial forces, bending stresses in two directions, worst case stresses, etc.

- You can create shear and moment plots. Right-click the **Results** folder and select **Define Beam Diagrams**. Plot types include: axial force, shear force in two directions, moments in two directions, and torque. You can use **Probe** to list the results for individual structural members.

Design Scenarios

Design scenarios have been enhanced. You can:

- Resize the dialog box to display more sets or parameters.
- Automatically fill in the values for a parameter in one of three methods: **Linear**, **Growth**, or **Distribute**. These methods are similar to those in Microsoft Excel. Right-click the parameter name and select **Series Fill**.
- Reset the values for each parameter or set to the current model value. Right-click the parameter or set name and select **Reset**.
- Copy tabular information to or from other Microsoft products such as Excel or Word. Select the table cells you want to copy or fill, then right-click and select **Copy** or **Paste**, respectively.
- Select a coordinate system for viewing results. Select **Choose Coordinate System** on the **Result Locations** tab of the **Design Scenario** dialog box.
- Display additional result types for design scenarios within static studies. These can be seen in the **Design Scenario Results Summary** and **Graph** dialog boxes. New types include: displacement in the X, Y, and Z directions (**UX**, **UY**, and **UZ**) and normal stresses in the X, Y, and Z directions (**SX**, **SY**, and **SZ**). These new types are available for selected vertices.

Large Displacement Support

The **Large displacement** option in static studies now works with:

- Cyclic symmetry restraints
- All spring connectors. Compression-only and extension-only springs were not supported in earlier releases.
- All bolt connectors

Trend Tracker (P)

Trend Tracker helps you detect trends in results from different iterations of a static study. After running a static study, you establish a baseline. Then you make changes to the geometry, loads, restraints, or any other feature and run the static study again. The software appends the new results as a new iteration.

To enable, right-click the study icon and select **Trend Tracker**.

Highlights include:

- A Trend Journal lists details about the baseline and every iteration.

- Graphs show the trend in important result quantities. You can view one of the default graphs or add a tracked data graph.
- A gallery can show displacement and stress plots for every iteration.
- Integration with Design Scenarios allows saving sets as iterations.
- A rollback function allows restoring the model to a specific iteration.

See **Trend Tracker** in the help.

Loads and Restraints

- The value box in the **Gravity** PropertyManager now includes a default value based on the units.
- You can apply mesh control to reference points. The mesher creates a node at every reference point. Loads applied at the specified reference points are applied at the exact locations.
- If you do not apply mesh control at reference points, you can still apply loads at reference points for static, frequency, and buckling studies. Since no nodes are necessarily present at these locations, the software applies the load to adjacent nodes. In earlier releases, you could apply point loads at vertices only. The reference points must lie within the model's boundary.

Bolt Connectors

Bolt connectors have been enhanced:

- (A) You can use bolt connectors in nonlinear analysis.
- You can simulate countersink screws and bolts.
- You can force the head and nut diameters to be equal.
- You can bolt more than 2 assembly components together. Select **Bolt series** under **Advanced Options** then select the cylindrical faces from the middle components. This option is available for static studies only.
- You can include the mass of the bolt in the analysis.
- If a plane of symmetry cuts through a bolt, you can define a symmetric bolt. Symmetric bolts work only for 1/2 or 1/4 symmetry. Select **Symmetrical bolt** under **Advanced Options**. For 1/2 symmetry, you also select the plane or planar face of symmetry. Symmetric bolts are available for static studies only.
- A **Grounded** bolt is now called a **Foundation Bolt**.

- When you add a bolt to one hole in a Hole Series, the software allows propagating bolts to all the other holes in the Hole Series.

It was possible in earlier releases to define bolts with **Split Lines** around the hole to select the head and nut contact faces. Now, you can only select the edges of the hole at the head and nut to define a bolt.

If you edit a bolt defined in an earlier release using head and nut contact faces, the program automatically selects the proper edges and enters the proper diameter values.

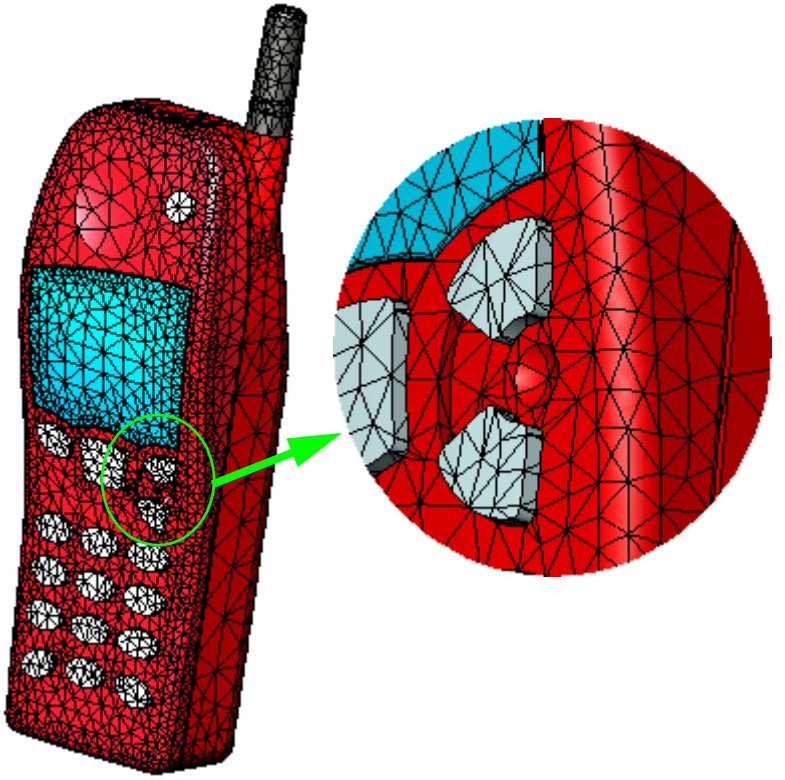
Pin Connectors

- Two Pin connector options have been renamed. **No Translation** has become **With retainer ring (No translation)**, and **No Rotation** has become **With key (No rotation)**.
- You can include mass information for pin connectors. Select **Include Mass** under **Advanced Options** and type the mass value. The mass value is used in frequency, buckling analysis, and in static and nonlinear analysis when gravity and centrifugal loads are selected.

Mesh

- You can mix beams, shells, and solids in a mixed mesh study.
- You can apply mesh control to reference points. The mesher creates a node at every reference point.
- In earlier releases, you had to remesh the model after you change a contact condition to use **Node to surface,** or **Surface to surface**. In this release, the mesh does not get invalidated because of such a change, and you do not need to remesh the model.
- The alternate mesher used in earlier releases is replaced by a curvature-based mesher. The mesher works for solid parts and assemblies in which incompatible bonding is used. It generates a finer mesh on areas of high curvature and transitions the mesh automatically. The following images show a model meshed by the new mesher.

To use the new mesher, click **Options** in the **Mesh** PropertyManager, select **Alternate**, and click ✓.



Contact and Bonding

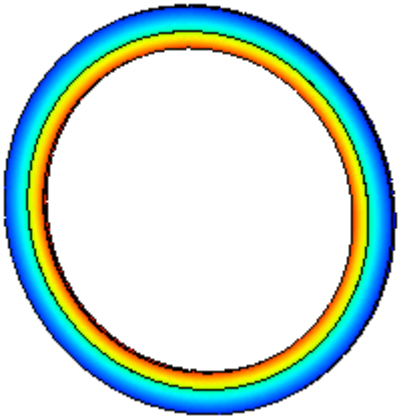
- In earlier releases, bonding face to face or edge to face with incompatible meshes used a node to surface bonding algorithm. This method produces inaccurate stresses in and around the bonded regions in some cases.

  In this release, the software uses a new algorithm based on the Mortar method for bonding faces to faces and edges to faces. The new algorithm gives continuous and more accurate stresses and bonding forces. It also accelerates convergence when using the h-adaptive method.

  To use the new method, select **Improve bonding with incompatible mesh (slower)** in the **Static**, **Buckling**, or **Frequency** dialog box before running the study.

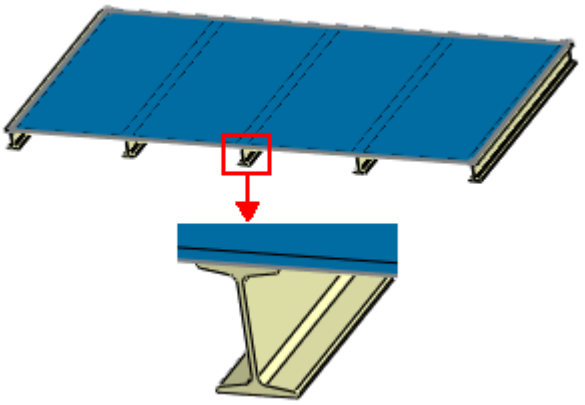
- The algorithm for the **Surface to surface** option for the **No penetration** and **Shrink fit** contact options is improved for the small displacement solution of static studies. The algorithm gives more accurate results for stresses and contact forces, and accelerates convergence when using the h-adaptive method.

  The figure shows von Mises stresses for a shrink fit of two rings.



Stiffeners

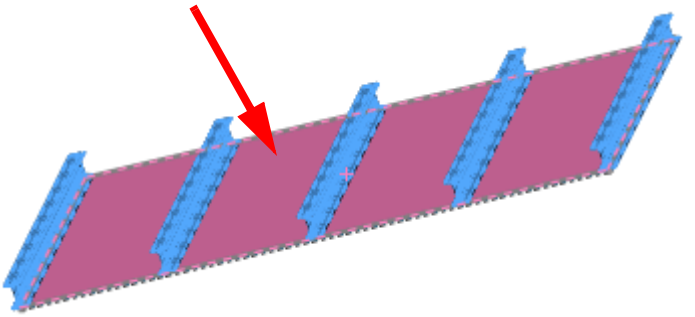
You can define beams as stiffeners on shell and solid faces.



To define a stiffener:

1. Create a mixed mesh study.
2. Define beams, shells, and solids.
3. Right-click the **Contact/Gaps** folder and select **Define Contact Set**.
4. In the PropertyManager, select **Bonded** under **Type**.
5. Click **Beams** and select the beams for the stiffener.

6 In **Faces for target**, select the face of contact, shown below, to be stiffened by the beams.



7 Click ✔.

Result Viewing

- (P) Time history graphs have been enhanced for nonlinear and drop test studies. In earlier releases, you could only use **Time** for the X-axis. In this release, you can use **Time**, **Stress**, **Displacement**, **Translational Velocity**, or **Acceleration** for the X-axis. For example, you can graph resultant displacements versus von Mises stress after running a nonlinear or drop test study.
- (A) In earlier releases, you could only list total strains for nonlinear studies. In this release, you specify the type of strain that you want to plot or list. The available types are:
  - Total (combined strains)
  - Elastic (recoverable strains)
  - Plastic (nonrecoverable strains)
  - Thermal (strains due to thermal loading)
  - Creep (strains due to creep effects)
- (P) You can view the change in results over several iterations of a static study. See **Trend Tracker (P)** on page 11-13.

Design Insight Plots

The new Design Insight plot shows the regions of the model that carry the loads most efficiently. Some users may recognize this plot as a “load path” plot. You can use this information to reduce the model’s material.

See **Design Insight** in the help.

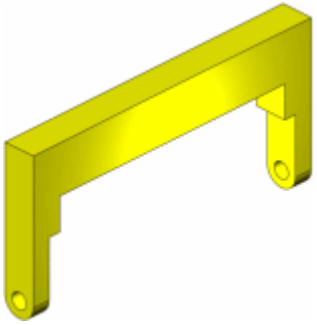
To use a Design Insight plot:

1. Open **COSMOSWorks\DesignInsight.sldprt**.
2. On the COSMOS AnalysisManager tree, right-click **Study 1** and select **Run**.
3. When analysis is complete, right-click the **Results** folder and select **Define Design Insight Plot**.

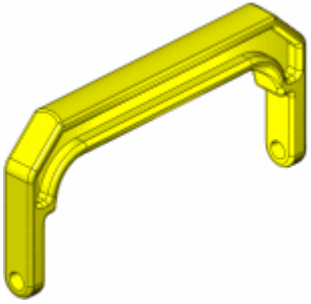
4. In the PropertyManager, move the slider to the right until the plot looks approximately as shown. Click ✓.

   The areas shown in blue indicate the areas that efficiently carry the load. The translucent areas of the plot indicate the boundary of the original model.

   You can use this Design Insight plot to update the model as shown. Toggle configurations in the sample part to see the differences. Notice how material was manually removed by a designer from regions that did not carry much of the load.



Before



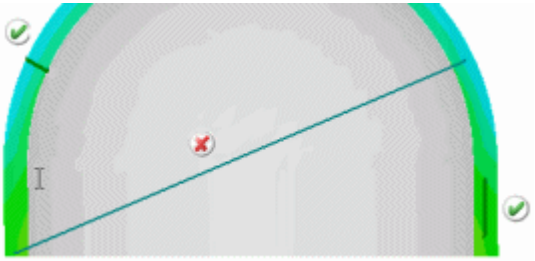
After

Stress Linearization (P)

In a stress field, it is useful to separate constant membrane and bending stresses. Such linearization is recommended by the American Society of Mechanical Engineers (ASME) International Boiler and Pressure Vessel Code.

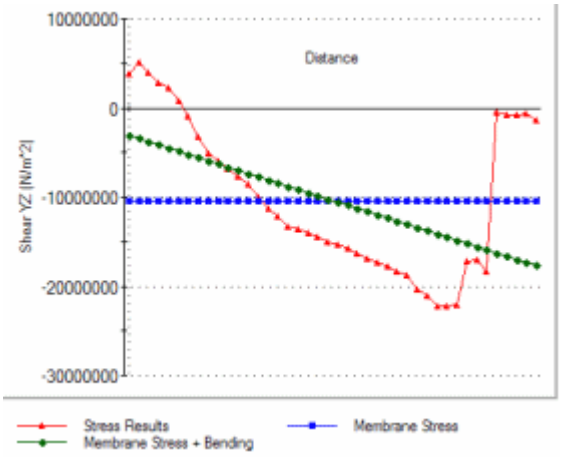
After defining a section plot for any stress component in a pressure vessel study, right-click the stress plot and select **Linearize**. You then define a line by selecting two locations on the section and click **Calculate** to list membrane, bending, and total stresses at the two locations.

The line connecting the two locations must fully lie on the material. It may not pass through holes or areas where results do not exist. For example, the long line in the figure below is not valid while the two other lines are valid. It is recommended to use a line normal to the boundaries as shown by the line on the left side. Lines not normal to the boundary, as the vertical line below, are not recommended.



To graph the results, click **Plot** in the **Linearize Stress** PropertyManager. The software generates 6 graphs. Each graph depicts the variation of membrane, bending, and total stresses of a component along the line. The six components are SX, SY, SZ, TXZ, TXY, and TYZ.

The X, Y and Z directions are defined by the line as explained in the **Linearize Stress** PropertyManager.



12

Other Functionality

This chapter describes enhancements to functionality of the software in the following areas:

- Installation
- Application Programming Interfaces
- DFMXpress
- DriveWorksXpress
- eDrawings
- Import/Export
- SolidWorks Explorer
- Mold Design
- Sheet Metal
- SolidWorks Rx
- Weldments

Installation

The SolidWorks Installation Manager combines installation, service packs, modification, and removal of all products. The Installation Manager lets you install from:

- SolidWorks media
- A source directory containing the SolidWorks 2008 files
- SolidWorks Download Center

Your serial numbers streamline the installation to default to the products you have purchased.

You can install immediately or schedule the Installation Manager to:

- Download and/or install at a later time
- Check regularly for updates and notify you when they are available

Application Programming Interfaces

Click **Help**, **API Help** to access the SolidWorks Application Programming Interface (API) Help systems.

The major enhancements made to the SolidWorks API in SolidWorks 2008 are:

- **Keywords**. Add your application's own keywords to SolidWorks files using the ModelDocExtension::AddOrUpdateSearchData method.
- **Undo commands**. Undo your application's commands using the ModelDocExtension::SetAPIUndoObject method and the SwUndoAPIHandler interface.
- **Step files**. Open STEP files using the SldWorks::GetImportFileData method, the ImportStepData::MapConfigurationData property, and the SldWorks::LoadFile4 method.
- **Annotation views**. Traverse annotation views using:

  Annotation::AnnotationView property

  ModelDocExtension::AnnotationViews property

  ModelDocExtension::IGetAnnotationViews method

  ModelDocExtension::AnnotationViewCount property

  ModelDocExtension::InsertAnnotationView method

  AnnotationView::Activate method

  AnnotationView::ActivateAndReorient method

  AnnotationView::AngleMadeWithViewHorizontal property

  AnnotationView::AnnotationCount property

AnnotationView::Annotations property

AnnotationView::GetHorizontalDirection method

AnnotationView::Hide method

AnnotationView::IGetAnnotations method

AnnotationView::InvertHorizontalDirection property

AnnotationView::MoveAnnotations method

- **Measure tool**. Measure distance, angle, radius, and size of and between lines, points, surfaces, and planes in sketches, 3D models, assemblies, or drawings using the Measure interface.
- **Triad manipulators**. Assign color to and specify how many circles are shown in a triad manipulator using the TriadManipulator::SetColorRefAtIndex method.
- **Keystroke events**. Intercept and pass keystroke events to your application using the PropertyManagerPage2Handler5::OnKeystroke method.
- **Pop-up menus and selection boxes**. Display a pop-up menu when a user right-clicks on a selection box in your application's PropertyManager pages using the PropertyManagerPageSelectionbox::AddMenuPopupItem method.
- **Manipulators**. Select manipulators, including drag arrows and triad manipulators, using PropertyManager pages because manipulators can be selected.
- **Mouse-wheel slider controls**. Create mouse-wheel slider controls on PropertyManager pages using the PropertyManagerPageNumberbox::SetRange2 and PropertyManagerPageNumberbox::SetSliderParameters methods.
- **PropertyManager page labels**. Allow rich text format in PropertyManager page labels using:

  PropertyManagerPageLabel::Bold property

  PropertyManagerPageLabel::CharacterBackgroundColor property

  PropertyManagerPageLabel::CharacterColor property

  PropertyManagerPageLabel::Font property

  PropertyManagerPageLabel::Italic property

  PropertyManagerPageLabel::LineOffset property

  PropertyManagerPageLabel::SizeRatio property

  PropertyManagerPageLabel::Underline property

- **PropertyManager message boxes**. Allow a message box to be placed anywhere (i.e., not just at the top) on a PropertyManager page by creating a group box containing just a label using the PropertyManagerPageGroup::BackgroundColor property and the previously listed PropertyManagerPageLabel properties.

- **Pictures on sketches**. Add and edit pictures on sketches using the SketchPicture interface and the Sketch::GetSketchPictureCount, Sketch::GetPictures, and SketchManager::InsertSketchPicture methods.
- **Silhouette edges**. Access silhouette edges in drawings using the SilhouetteEdge interface and the SelectionMgr::GetSelectedObjectsFace method.
- **Drawing views**. Traverse drawing views on sheets more easily, and not activate those sheets, using Feature::GetSpecificFeature2 and Sheet::GetViews methods.
- **Centermarks**. Get and set the length of the arms of annotation-based centermarks using:

  CenterMark::IsGrouped property

  CenterMark::GroupCount property

  CenterMark::GetExtendedLength method

  CenterMark::SetExtendedLength method
- **Lofts**. Create temporary loft bodies using the Modeler::CreateLoftBody method, and create thickened sheets for temporary thickened bodies using the Modeler::ThickenSheet method.
- **Derived part features**. Create derived part features using the DerivedPartFeatureData interface.
- **Hole series**. Access hole series using the HoleSeriesFeatureData interface.
- **Break corner features**. Add corner cuts centered relative to the bend lines on sheet metal break corners using the BreakCornerFeatureData::CenteredOnBendLines property. Add material to the internal corners of sheet metal break corners using the BreakCornerFeatureData::InternalCornersOnly property.
- **Closed corner features**. Specify gap distances, overlap/underlap ratios, and whether to use open bend regions for sheet metal closed corners using the ClosedCornerFeatureData::GapDistance, ClosedCornerFeatureData::OverlapUnderlapRatio, and ClosedCornerFeatureData::OpenBendRegion properties.
- **Gauge tables and base flanges**. Specify gauge tables for sheet metal base flanges using:

  BaseFlangeFeatureData::GaugeTablePath property

  BaseFlangeFeatureData::GetTableRadii method

  BaseFlangeFeatureData::GetTableRadiiCount method

  BaseFlangeFeatureData::GetTableThicknesses method

  BaseFlangeFeatureData::GetTableThicknessesCount method

  BaseFlangeFeatureData::KFactor property

BaseFlangeFeatureData::OverrideKFactor property

BaseFlangeFeatureData::OverrideRadius property

BaseFlangeFeatureData::OverrideThickness property

BaseFlangeFeatureData::TableKFactor property

BaseFlangeFeatureData::TableRadius property

BaseFlangeFeatureData::TableThickness property

BaseFlangeFeatureData::ThicknessTableName property

BaseFlangeFeatureData::UseGaugeTable property

- **Multibodies**. Specify which body to keep after cutting a body into two bodies in parts and assemblies using the Feature::SetBodiesToKeep method and the PromptBodiesToKeepNotify events.
- **Mirror pattern features**. Specify feature scope for mirror pattern features using:

  MirrorPatternFeatureData::FeatureScope property

  MirrorPatternFeatureData::FeatureScopeBodies property

  MirrorPatternFeatureData::GetFeatureScopeBodiesCount method

  MirrorPatternFeatureData::IGetFeatureScopeBodies method

  MirrorPatternFeatureData::ISetFeatureScopeBodies method
- **Fillets**. Trim and attach fillets, including surface fillets, using the SimpleFilletFeatureData2::TrimAndAttachSurfaces property.
- **PhotoWorks scenes**. Access PhotoWorks scenes (materials, decals, and textures) using the Decal, FaceDecalProperties, and RenderMaterial interfaces and these methods:

  Component2::GetDecals method

  Component2::GetDecalsCount method

  Component2::GetRenderMaterials method

  Component2::GetRenderMaterialsCount method

  Face2::GetAllDecalProperties method

  Face2::GetDecalsCount method

  Face2::IGetDecalProperties method

  ModelDocExtension::AddDecal method

  ModelDocExtension::AddDefaultRenderMaterial method

  ModelDocExtension::AddRenderMaterial method

  ModelDocExtension::CreateDecal method

  ModelDocExtension::CreateRenderMaterial method

  ModelDocExtension::DeleteAllDecals method

ModelDocExtension::DeleteDecal method

ModelDocExtension::DeleteRenderMaterial method

ModelDocExtension::GetDecal method

ModelDocExtension::GetDecals method

ModelDocExtension::GetDecalsCount method

ModelDocExtension::GetKeepLightInRenderScene method

ModelDocExtension::GetLightEnabledInRender method

ModelDocExtension::GetMaterial method

ModelDocExtension::GetRenderMaterials method

ModelDocExtension::GetRenderMaterialsCount method

ModelDocExentsion::HideDecal method

ModelDocExtension::InsertScene method

ModelDocExtension::MoveDecal method

ModelDocExtension::ReverseDecalsOrder method

ModelDocExtension::SetKeepLightInRenderScene method

ModelDocExtension::SetLightEnabledInRender method

ModelDocExtension::UpdateRenderMaterialsInSceneGraph method

- **Motion studies**. Create animations and presentations and simulate the basic and complex effects of physics on assemblies using the MotionStudyManager and MotionStudy interfaces.
- **Visual properties**. Being enhanced to work with the new SolidWorks RealView graphics.

See the *SolidWorks API Release Notes*, which is available in the Help, for a list of all of the changes made to the SolidWorks API 2008.

DFMXpress

Use DFMXpress to check your designs for manufacturability.You can identify areas that are difficult, expensive, or impossible to machine early in the design process.

Rule Checks

DFMXpress lets you validate parts against these design rules:

- **Hole Depth/Diameter Ratio**: Checks for holes that are too deep or have too small a diameter for efficient chip removal.
- **Inaccessible Features**: Checks for inaccessible features that could require special cutting tools or procedures.
- **Linear and Angular Tolerance Rule**: Checks for stringent tolerances that might not accommodate standard process parameters and tools.
- **Mill Sharp Internal Corners**: Checks that corners are not too sharp for standard milling operations.
- **Partial Holes**: Checks that holes that break through the boundary of the part are a specified percentage within the material.
- **Bore Relief - For Turned Parts**: Checks that blind bored holes have relief at their ends.
- **Deep Pocket/Slot**: Checks that slots are not too deep and narrow for standard cutters.
- **Hole Entry/Exit Surface**: Checks that the entry and exit surfaces of drilled holes are perpendicular to their axes so that drill tips do not wander or produce uneven exit burrs.
- **Holes with Flat Bottom**: Checks that blind holes are cone-bottomed rather than flat-bottomed to facilitate drilling and reaming.
- **Minimum Corner Radius - For Turned Parts**: Checks that corners accommodate tools with a large nose radii.
- **Conform to standard drill sizes**: Checks that hole sizes match standard drill sizes.
- **Fillets on Outside Edges**: Checks that chamfers rather than radii are specified for top face boundary edges.
- **Hole Intersects Cavity**: Checks for drilled holes that intersect with cavities.

To check parts:

1. Open a part.
2. Click **Tools**, **DFMXpress** .
3. Click **Run**.

   The results list **Rules Failed** and **Rules Passed**.

4 Expand the **Rules Failed** tree to view the rules that generated failures.

5 Expand the rules to view failure instances.

6 Select an instance.

A tooltip explains the failure and the problem feature is highlighted in the graphics area.

Hands-on Example

Rule Configuration

You can set these parameters to determine how rules are evaluated:

- **Hole depth to diameter ratio**
- **Minimum % of hole area inside part**
- **Mill tool depth to diameter ratio**
- **Minimum corner radius (Turn Part)**
- **Minimum % of bore relief (Turn Part)**
- **Minimum linear tolerance zone**
- **Minimum angular tolerance zone**

To configure parameters:

1 In the DFMXpress pane, click **Settings**.

2 Select **Prismatic** or **Turned** for **Part type**.

3 Select values for specific parameters.

4 Click **Close** to save changes.

DriveWorksXpress

DriveWorksXpress is a design automation tool you can use to automatically generate parts, assemblies, and drawings based on predefined design information.

Hands-on Example

To open DriveWorksXpress:

Click **Tools**, **DriveWorksXpress** .

Overview

Using DriveWorksXpress includes these general steps:

- Capturing (identifying) the dimensions and features of the model you want to vary.
- Setting up custom properties.
- Linking a drawing to the model to generate drawings along with the model variations.
- Creating a form for data input.
- Defining rules to apply to the entered data.
- Running the model to generate variations.

Capturing Parameters

You capture the features, dimensions, and folders you want to vary in the model to generate new variations. Captured parameters are stored in the DriveWorksXpress database.

Creating Custom Properties

Custom properties add information to the model that you can use, for example, for drawing borders or annotations. The properties you add through DriveWorksXpress are exactly the same as properties you add to the model manually.

Designing Input Forms

You design forms to collect the data needed to generate new versions of the model. DriveWorksXpress provides these types of controls for form building:

- **Text Box**
- **NumericText Box**
- **Spin Button**
- **Drop Down**
- **Check Box**

A test feature lets you try out each control.

Creating Rules

Rules are applied to the form input to drive the captured parameters and generate new models.

You can create the following **Rule Types**:

- **File Names**: Append text to the file name. For example, the text entry for **Customer** on the input form could be appended to the file name, transforming Block.sldasm to Block Acme.sldasm.
- **Configurations**: Switch to any existing configuration within a file.
- **Custom Properties**: Set the value for the custom property. For example, selecting the name of Joe Doe from a **Drawn By** drop-down list in the input form sets the custom property to Joe Doe.
- **Dimensions**: Create a formula to set the value of the dimension.
- **Features**: Change the state of the feature to suppressed, unsuppressed, or delete.

Create rules in the build window on the **Rules** tab using Microsoft Excel syntax.You can enter text directly or use shortcuts in these tabbed sections:

- **Inputs**: Select controls from the input form, for example, the text entered for **Drawn By**.
- **Recent**: Select entries you have created as **Change Quick Text**.
- **Math**: Select mathematical operators, for example, +, -, &.
- **Logic**: Select logical operators, for example, =>, NOT( ), IF (, ,).

Running Models

Models are ready to run when the form and rules are designed and tested. The Welcome page indicates how many of your models are **Ready to Run**.

Run a model to create a new model based on the data you enter on the input form.

eDrawings

Drawings

You can change custom properties and revision table data in SolidWorks drawings without opening the drawings in SolidWorks.

- In Microsoft Windows Explorer, right-click and select **Properties**.
- In SolidWorks Explorer, select a drawing file in the local view and modify properties on the **Properties** tab .

Write access is required. You can change properties without changing the revision number. Previews in eDrawings, SolidWorks, and SolidWorks Explorer reflect the new data.

Mozilla Firefox

In Mozilla® Firefox®, you can open HTML files that reference external eDrawings files. (You cannot open HTML files saved from eDrawings.)

Previews

The eDrawings Viewer **Open** dialog box displays previews of eDrawings files, SolidWorks files, and all types of files published to eDrawings if the files have been saved in a 2008 application.

Pro/ENGINEER Files

Pro/ENGINEER models can be imported with the following limitations:

- Parts and assemblies can be imported, but drawings cannot.
- If a model uses Pro/ENGINEER family table instances, then the XPR and XAS accelerator instance files must be included for correct results.
- Colors in part files are recognized, but colors applied at the assembly level are not.
- Certain Pro/ENGINEER features, such as annotations, sketches, and layers, are not imported.

RSS Feed

The eDrawings Viewer includes an RSS (Really Simple Syndication) feed in the status bar. The RSS feed contains information about eDrawings, such as available service packs, or about related products.

- To hide the RSS feed, hide the status bar.
- To view details on the current information, click in the RSS feed in the status bar.
- To view all RSS feeds, click .

SolidWorks Display States

The eDrawings Viewer shows SolidWorks-created display states on the **Configurations** tab, in a section labeled **Display States**. Select a listed state to display it in the graphics area.

If **Linked Display State to Configuration** is selected in SolidWorks, each configuration remembers the last display state and previews use the display state corresponding to the previewed configuration.

eDrawings 2008 supports display states for assemblies published from SolidWorks 2006 and 2007 with the following limitations:

- Texture attributes are not shown.
- The configuration name is prepended to the display state name.
- The list of display states changes when the configuration changes because display states were tied to a configuration.
- The configuration current when the file was published cannot be determined.

STL (stereolithography) Files

You can import STL (***.stl**) files into eDrawings.

XPS (XML Paper Specification) Files

You can generate XPS files (***.edrwx**, ***.eprtx**, ***.easmx**) with **Save As** in SolidWorks and open them in eDrawings or in the XPS Viewer.

The XPS Viewer is included in the Vista operating system and is distributed with .NET Framework 3.0, which is installed with SolidWorks on XP and Server 2003 operating systems.

Import/Export

Adobe Illustrator

The Adobe® Illustrator® ***.ai** file type is available for output with **Save As**. Part and assembly models are saved as images.

You can copy and paste from Adobe Illustrator into SolidWorks sketches and drawings, and vice versa.

Adobe Photoshop

SolidWorks data (sketches, parts, assemblies, and drawings) can be saved as Adobe Photoshop® ***.psd** files with **Save As**. SolidWorks data saved as Adobe ***.pdf** files are also accepted in Photoshop.

The Photoshop ***.psd** file type is available for import on the **File**, **Open** menu. Single-layer and multi-layer images are inserted as SolidWorks Sketch Pictures. For multi-layer Photoshop files, you select which layers to import and the layers are merged into a single image.

Photoshop files can be used as image backgrounds in SolidWorks. Photoshop file types are available in the **Decals** menu for images and masks and the **Materials** menu.

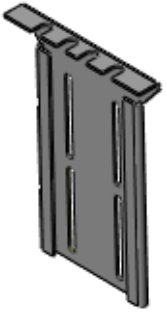
Autodesk Inventor

You can import Autodesk® Inventor® assembly files (***.iam**) into SolidWorks with **File**, **Open**. You must have Inventor installed on your desktop, but it does not have to be running, to use the Inventor translator. You can now import files from Inventor 11 and above.

DXF/DWG Mapping

The dialog box for exporting SolidWorks custom map properties to DXF/DWG files is on one page for access to layers, map entities, and map colors.

Sheet Metal Flat Patterns



You can create ***.dxf** files of sheet metal flat patterns directly from sheet metal part documents without flattening the model.

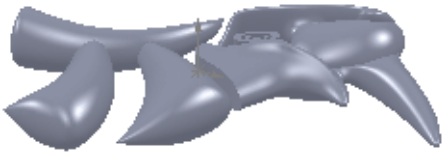
- Click **File**, **Save As** and select **DXF (*.dxf)** for **File of type**.
- Right-click **Flat-Pattern** in the FeatureManager design tree and select **Export Flat Pattern to DXF/DWG**.

The words **Flat pattern** are prepended to the file name.

Pro/ENGINEER

To import Pro/ENGINEER® assembly constraints, select **Import component constraints** in the **Pro/ENGINEER to SolidWorks Converter** options dialog box. The Pro/ENGINEER constraints are translated into SolidWorks assembly mates.

Rhino



Importing Rhino® (***.3dm**) multibody files results in one SolidWorks part file.

Import options allow you to specify how surfaces and solids on hidden Rhino layers are handled.

Rhino is integrated into SolidWorks menus for actions such as **Edit Feature** and **Insert**, **Features**, **Imported**.

XPS (XML Paper Specification) Files

You can generate XPS files (***.edrwx**, ***.eprtx**, ***.easmx**) with **Save As** in SolidWorks and open them in eDrawings or in the XPS Viewer. XPS is an Add-In in SolidWorks and is selected by default.

The XPS Viewer is included in the Vista operating system and is distributed with .NET Framework 3.0, which is installed with SolidWorks on XP and Server 2003 operating systems.

SolidWorks Explorer

Right Pane Previews

When you open a folder, you can navigate through the files in the right pane using a new display mechanism.

To display previews:

1. Click on a document in the file list.
2. Use the horizontal scroll bar below the preview to scroll through the documents.

Tags

Tags let you associate keywords with documents to make it easier to search for them.

To tag a document in SolidWorks Explorer:

1. Open SolidWorks Explorer.
2. Select a document on the File Explorer tab in the left pane, or on the file list in the right pane.
3. In the **Tags** field at the bottom of the left pane, click to display a list of tags that are already in use.
4. Select a tag from the list or type a new tag.

To use a tag in a SolidWorks Explorer search:

In the **Search** box at the top of the left pane, type the tag, and press **Enter**.

The **Results** tab displays the documents that you tagged, with the tag identified as a DocKeyword.

Mold Design

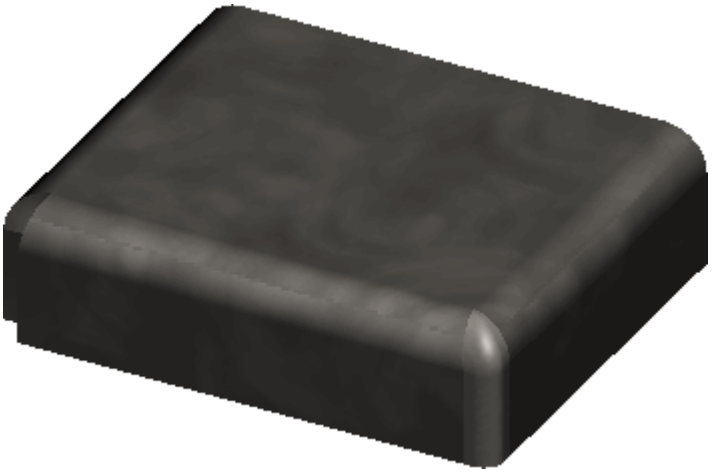
MoldflowXpress

MoldflowXpress is no longer available.

Sheet Metal

Adding Weld Beads to Sheet Metal Parts

You can add weld beads to the corners of a folded sheet metal part for better graphic representation in an as-welded state. When you flatten the part, the weld beads are suppressed.



Hands-on Example

Exporting Flat Patterns

The flat pattern of a sheet metal part can be exported as a DXF file without having to create a drawing first. See **Sheet Metal Flat Patterns** on page 12-13.

SolidWorks Rx

In Microsoft Windows, click **Start**, **SolidWorks 2008**, **SolidWorks Tools**, **SolidWorks Rx** .

In SolidWorks Rx, new features are available from the **Home** tab:

- **File & Logs** tab where you can now open Rx log files.
- **Addins** tab where you can manage and run external utilities.
- SolidWorks Safe Modes:
  - **Start SolidWorks in Software OpenGL mode** to pinpoint issues with a graphics card.
  - **Bypass the Tools/Options settings** for the next session of SolidWorks.

A new error recovery option, **Report the problem (Start SolidWorks Rx)**, is now available on the **Restarting SolidWorks page**.

Video is now captured based on SolidWorks events. You can set the **Video Recording Scale** and **Video Quality** and **Limit Capture Events**.

Weldments

Transferring Cut List Information with Weldment Body

When you insert a weldment body from a weldment part into a new part, and then convert the new part to a weldment part:

- Cut list information for the inserted body appears in the cut list folder of the new part.
- Changes to cut list information in the parent part update the derived part.
- If the link is broken between the parent and derived part, the cut list information is preserved in the derived part.

Bills of Materials

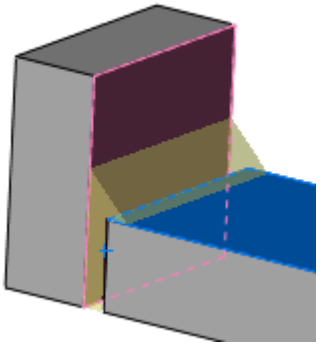
- You can expand weldment parts in an assembly's Bill of Materials to list the unique structural member profiles needed to produce the part. The total length required for each profile appears in the quantity column.
- You can dissolve weldment parts so that only the total material requirements appear in the assembly's Bill of Materials. When you dissolve multiple weldments, like items are combined.

See **Weldment Material Usage** on page 9-5.

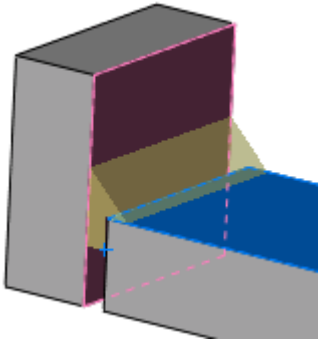
Welding Bodies With Gaps

When you insert a weld bead across a gap between the faces to be welded, you can specify:

- Full penetration of the gap
- A distance for partial penetration of the gap



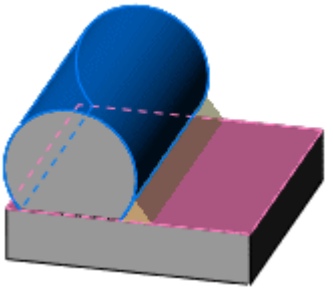
Full penetration



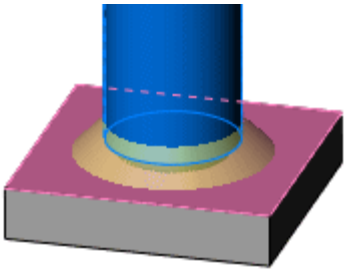
Partial penetration

You can also apply weld beads:

- Along cylinders



- Along toroidal faces



Structural Member Profile Orientation

When inserting a structural member, you can:

- Mirror the weldment profile about its horizontal or vertical axis.
- Align the horizontal or vertical axis of the profile with any vector you select.

13

SolidWorks Office Professional Components

This chapter describes enhancements to the SolidWorks Office Professional components in the following areas:

- FeatureWorks®
- PhotoWorks
- SolidWorks Design Checker
- SolidWorks Utilities
- PDMWorks® Workgroup
- SolidWorks Task Scheduler
- Toolbox

FeatureWorks

Automatic Feature Recognition

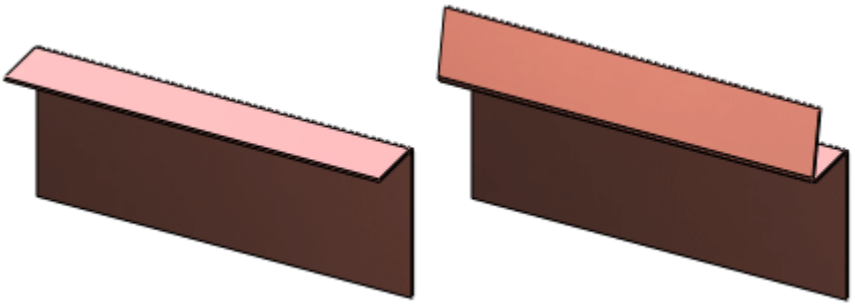
Edge flanges and hems have been added to **Automatic Features** in the **FeatureWorks** PropertyManager. Existing sheet metal limitations apply to both features.

See **FeatureWorks PropertyManager** in the help.

Edge Flanges

FeatureWorks recognizes:

- Simple edge flanges
- Multiple edge flanges
- Edge flanges created at the end of existing edge flanges

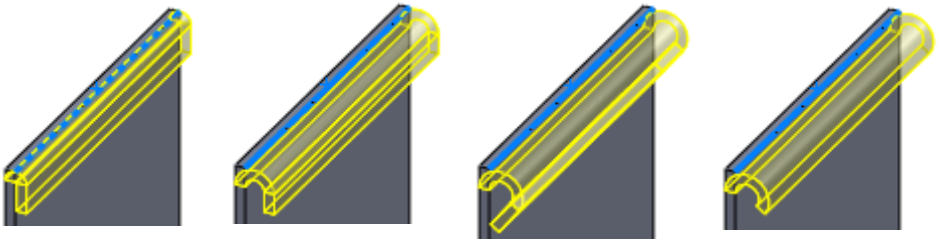


Recognition limitations include edge flanges:

- Created on split faces
- With an edge flange thickness that is less than the length of the model
- With modified lateral geometry
- With cuts

Hems

FeatureWorks recognizes the four types of hems supported in SolidWorks:



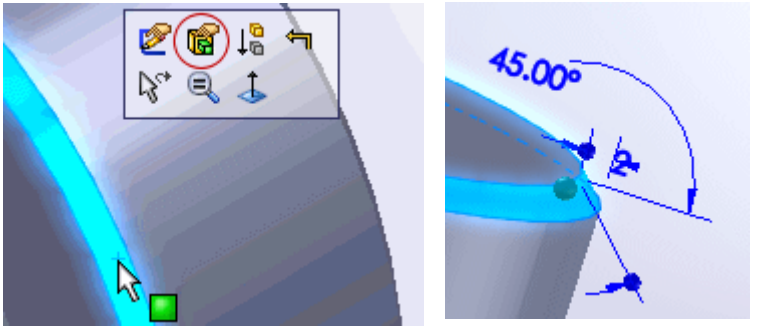
Closed Open Tear Drop Rolled

Recognition limitations include hems:

- With intersecting hem faces
- Created on split edges
- With overlapping contours (after the bends are flattened)
- With non-cylindrical faces for the face bend
- That merge into the base body
- With cuts on the flanges
- With a length that is less than the thickness

Resize Tool

You can select faces that includes a hole, a fillet or a chamfer feature to display the context toolbar. Click **Edit Feature** to display and edit the parameters of the feature in the graphics area or in the PropertyManager.



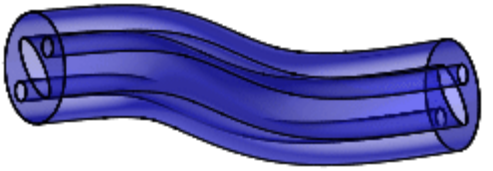
If the feature includes multiple instances that were created simultaneously, only the instance on the selected face is modified.

Once you edit the feature (if you have not used the **Recognize Feature** option), it adds the feature to the FeatureManager design tree below **Imported** .

See **Resize Tool** in the help.

Base Sweeps with Internal Loops

FeatureWorks has extended **Interactive** recognition to base sweeps with internal loops.



Base sweep with 3 internal loops

Recognition limitations include sweeps:

- With the **Orientation Type** set to any option other than **Follow Path**
- With guide curves
- With a closed path
- With drafts

PhotoWorks

Batch Document Processing

You can schedule batch processes to render PhotoWorks documents. To schedule batch processing:

1. Open the first document.
2. Click **Render to File** (**PhotoWorks** toolbar).

If the appearances you applied are not available, you can browse for a replacement by clicking **Yes** in the **PhotoWorks Render** dialog box. Click **No** to ignore all missing appearances.

3. Click **Schedule** in the **Render to File** dialog box.
4. In the **Render/Animation** dialog box, enter the required information.
5. Click **Finish** and:
   - Repeat steps 1 through 5 for each document you want to schedule or to render the same document with different appearances multiple times.

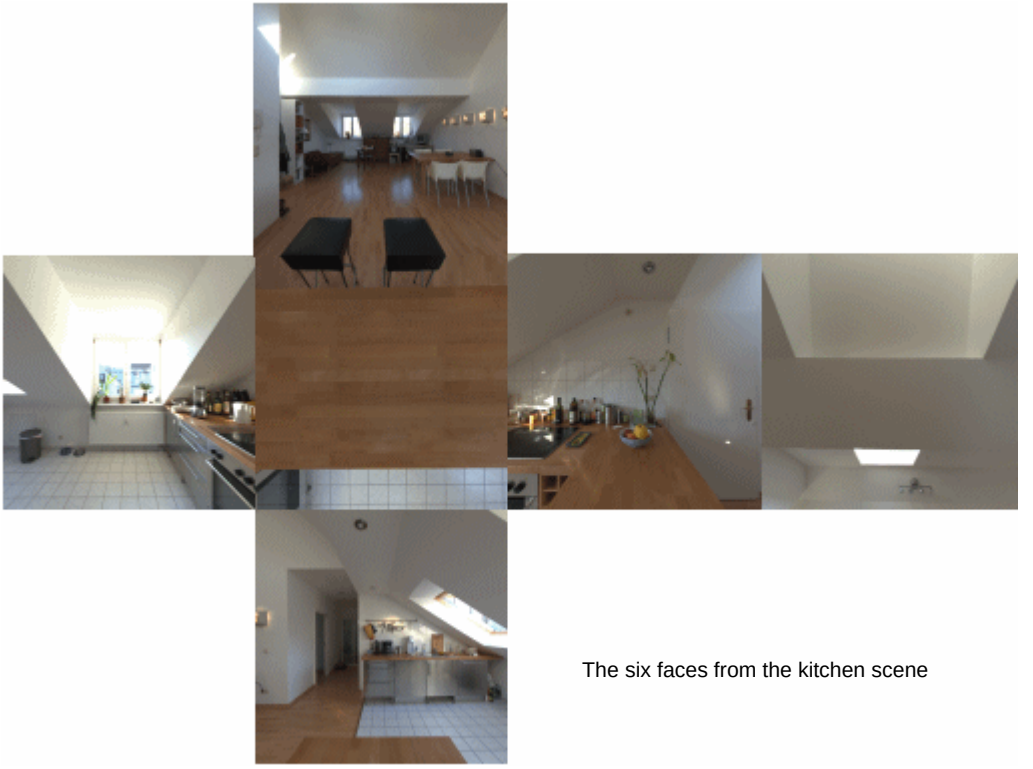
If a scheduled document is too complex for the system's available memory, batch processing skips that document and processes the next scheduled document.

See **Batch Document Processing** in the help.

As with other batch tasks, you can reorder tasks, generate reports, etc. through the Task Scheduler application. See **SolidWorks Task Scheduler** on page 13-20.

Cubic Environment Mapping

Cubic environment mapping captures reflections using the 6 projected faces surrounding an object. To establish a reflective environment, 2D images are mapped onto the 6 virtual walls of the environment.



The six faces from the kitchen scene

If you want to repeat an image on each wall, from the **Sene Editor**, select the **Environment** tab, and then select **Repeat image on each wall**.

Indirect Illumination

The controls for **Indirect Illumination** on the **Illumination** tab, **Options** (PhotoWorks toolbar), include pre-set or user-defined settings.

Pre-Set

Select from a range that includes: **Draft (default)**, **Low**, **Medium**, **High**, **Photographic**, **High Quality Photographic**, or **User Defined**.

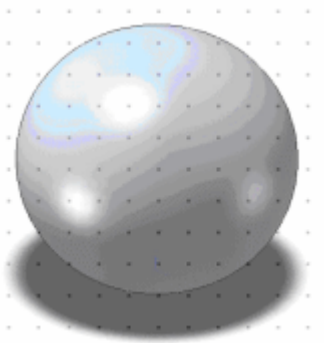
As you select each setting, average rendering speeds are displayed using the **Draft (default)** setting as a baseline.

Lights, appearances, and the options you set all affect rendering speed. Consequently, comparisons to the **Draft (default)** setting are relative indications.

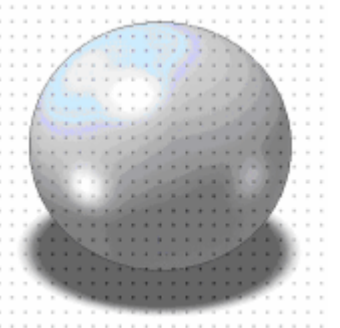
User Defined

Select **User Defined** for **Indirect Illumination** quality to adjust the following ranges:

- **Details**. Controls the density of light gathering points and determines how clearly you see rapid changes between light and dark values. High settings sharpen contrast, but increase the rendering time. Increasing the value for **Details** requires increasing the value for **Precision**.



Details (and Precision): Low value results in a minimal number of light gathering points.

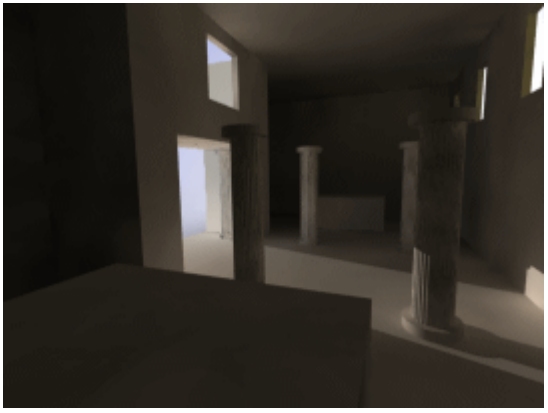


Details (and Precision): High value increases the number of light gathering points.

- **Averaging**. Controls the size of the area over which different sections in a rendering are blurred together. Low values display harder transitions between bright spots and less illuminated spots. Rendering time is marginally affected.

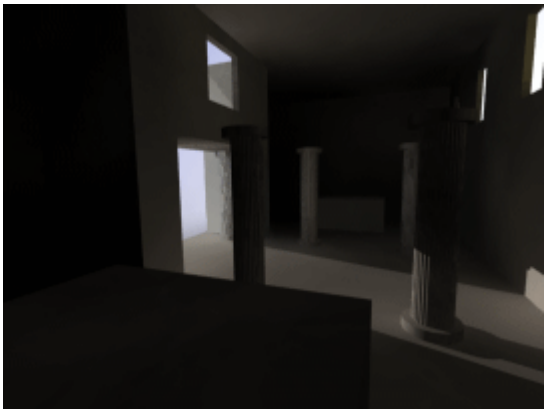
- **Precision**. Modifies the distance between successive calculations at light gathering points. Increasing the proximity between the points, requires more calculations. This heightens the photorealistic effect, but increases the rendering time.

Precision and Details set to high. Details are clear, despite low light level.



- **Bounces**. Determines how many times light is transferred from one surface to another. Increasing the value distributes more light; setting values too high makes colors bleed.

Bounce set to 2. Low value distributes light sparingly.



Appearances and Scenes

New appearances and scenes were added. Many existing appearances were re-calibrated. Documents created with **RealView** (SolidWorks application), and documents rendered with the PhotoWorks application now look more uniform.

> Although both applications use the same appearances and scenes, the PhotoWorks application adds greater flexibility with options not available in the SolidWorks application.

See **RealView Workflow** on page 2-2.

New and re-calibrated appearances include:

- Emissive appearances (see **Emissive Appearances** on page 13-10).
- Satin finishes.

New scenes (such as **Grill Lighting**) and appearances combine transparency with the capability to display shadows.

Appearances PropertyManager

The **Appearances** PropertyManager includes **Basic** and **Advanced** tabs. When you add the PhotoWorks application, you can choose to:

- Click the **Basic** tab to display the **Color/Image** and **Mapping** tabs. Use the **Basic** tab to add appearances with a simplified interface. See **RealView Workflow** on page 2-2.
- Click the **Advanced** tab to display four tabs: **Color/Image**, **Mapping**, **Illumination**, and **Surface Finish**. Use the **Advanced** tab to have greater control over appearances.

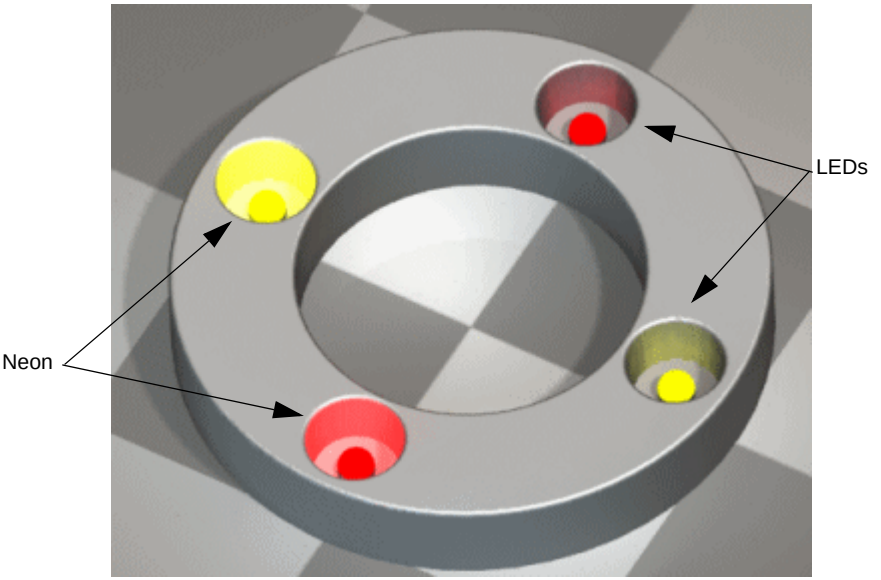
Emissive Appearances

From the **Appearances** folder, under **Lights**, you can represent low-level light emitters including:

- **LED** (light emitting diodes)
- **Neon Tube**
- **Backlit LCD**
- **Area Light** (light emitted from a region, such as studio light boxes)

To apply emissive appearances, follow these suggestions:

- Adjust the **Brightness** for **Directional**, **Spot**, or **Point light**.
- Add emissive entities to neutral, non-reflective areas (flat paints, non-shiny metals, etc.). Reflective appearances can diminish the effects of emissivity.
- Surround the geometry to give the light source faces off which to reflect.
- Apply lights to small entities.
- View emissive entities from short distances.

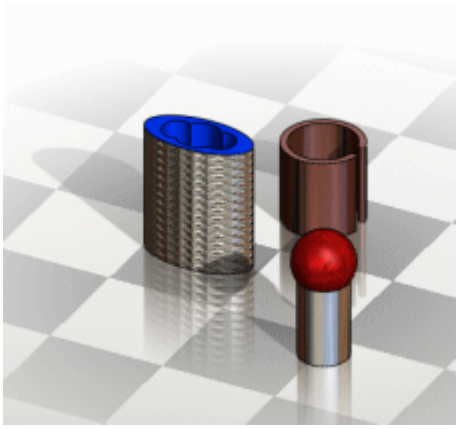


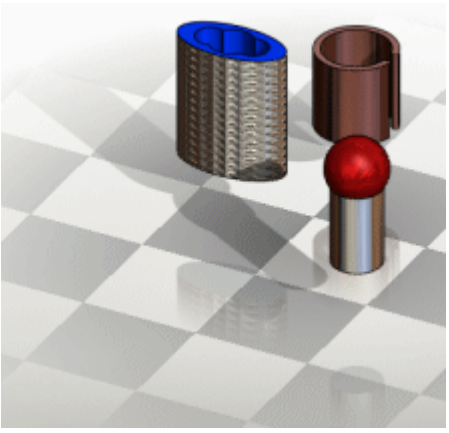
PhotoWorks Scene Editor

New controls in the **Scene Editor** were added to these tabs:

Room

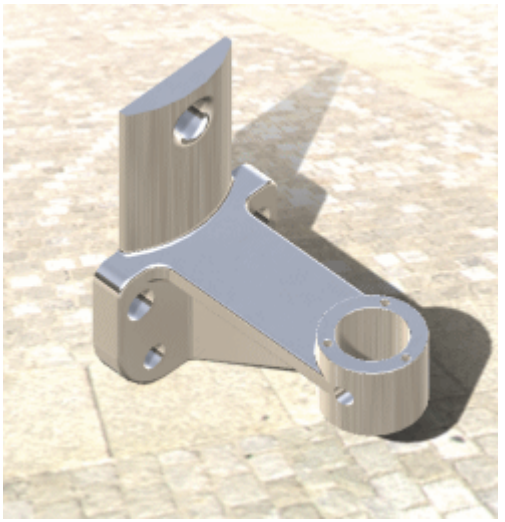
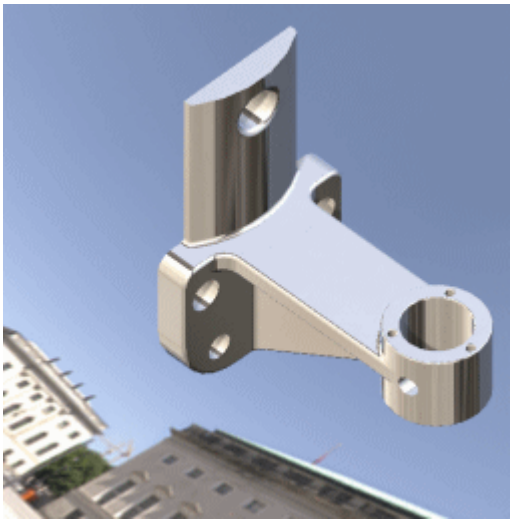
- **Floor offset**. Offsets the model geometry from the scene floor.





- **Floor Rotation**. Spins the floor.

- **Selected Planar Face** (Under **Align with**). Flips the floor about the selected model geometry based on the planar face you select.

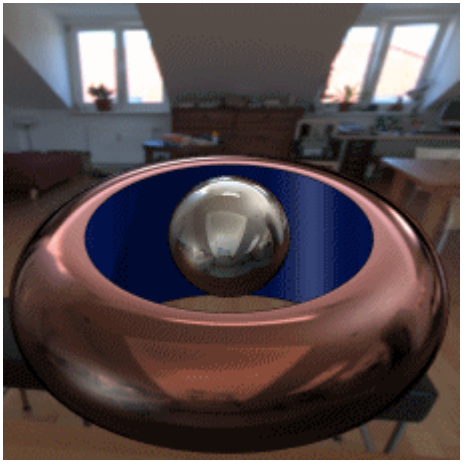
Rotate and position the second image to place it on the cobblestone floor.

Back/Foreground

Set a value for **Background Rotation** with **Spherical Image** or **Cubic Image** selected (see **Environment Rotation** below).

Environment

Environment Rotation. With **Spherical Image** or **Cubic Image** selected, set a value to rotate the background image about an axis that is normal to the floor.



See **Scene Editor** in the help.

System Options

Enable Memory Settings

Memory management was improved with a simplified user interface to facilitate rendering large models.

The controls for **Enable memory settings** on the **System Options** tab, **Options** (PhotoWorks toolbar), have replaced the multiple adjustments formerly located on the **Document Properties** tab.

Memory management benefits:

- Shorter rendering times for large models.
- Increased document size limits.
- User alerts when models are too complex for the system's resources.

Gamma Correction

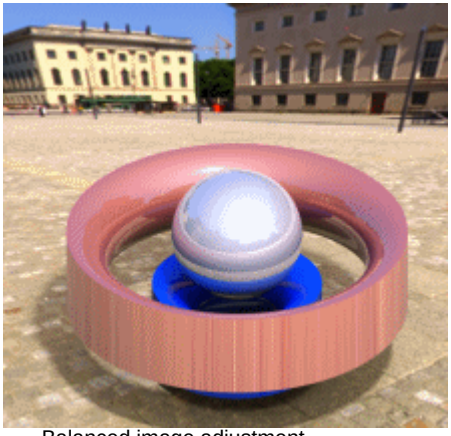
An image was added to facilitate monitor settings. The image updates as you change values for **Screen image gamma correction**.

Image Adjustment

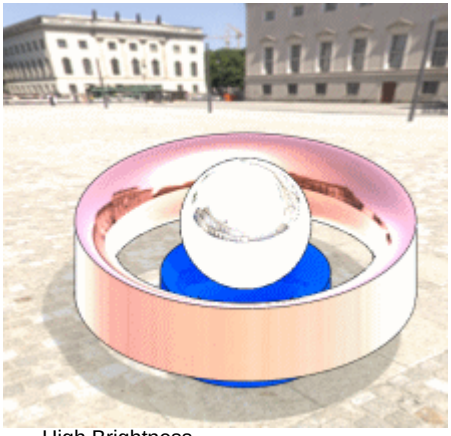
On the **Document Properties** tab, **Image Adjustment** controls tone mapping. Tone mapping accounts for high dynamic range images, and is synonymous with the types of adjustments found in most graphics' applications.

The settings for **Image Adjustment** include:

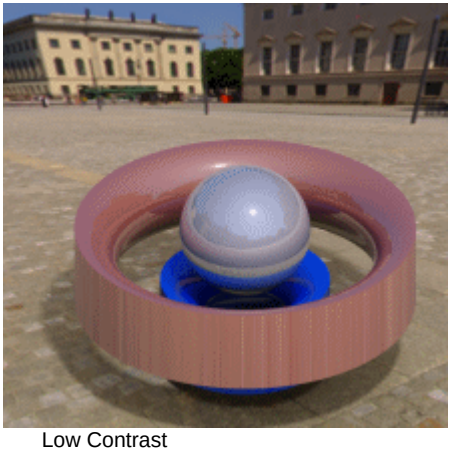
- **Brightness**. Controls the radiance of an image.
- **Contrast**. Controls the level of difference between light and dark areas.
- **Color Saturation**. Controls the level of color intensity. On most scenes, you can set **Color Saturation** to 100%. On some scenes, it makes the colors glow. If the setting is too low, the colors are washed out.



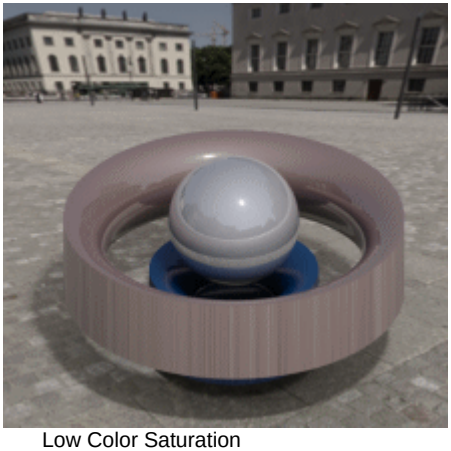
Balanced image adjustment



High Brightness



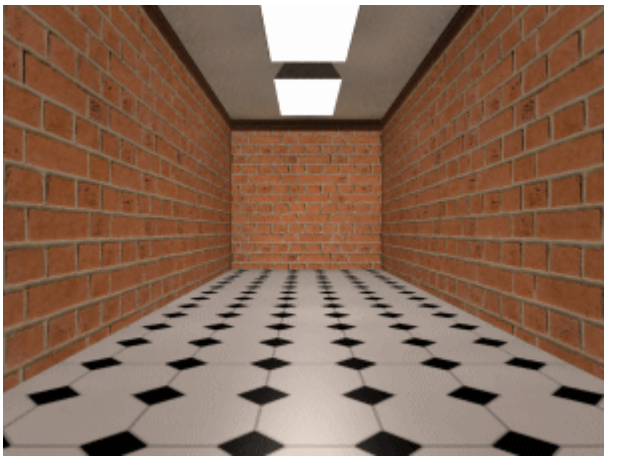
Low Contrast



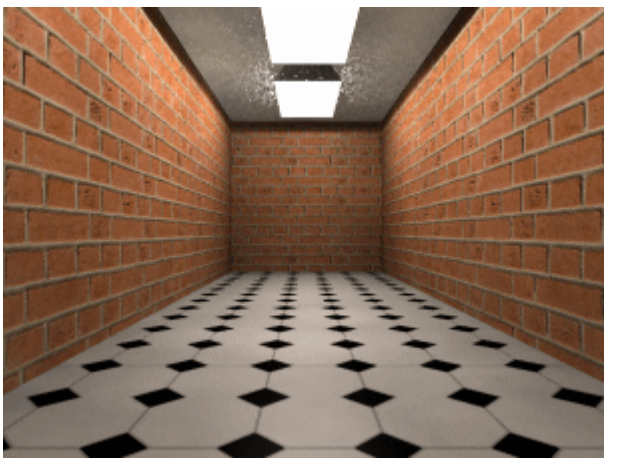
Low Color Saturation

Use Realistic Falloff

A toggle switch, **Use Realistic Falloff**, was added to the **Spot** , **Point** , and **Area** light PropertyManagers. When selected, it improves photo realism by controlling how light intensity diminishes as a function of distance.



Use Realistic Falloff cleared



Use Realistic Falloff selected

SolidWorks Design Checker

Check Documents from DWG Files

You can generate check documents based on the parameters in existing **DWG** files. Valid checks from **DWG** files include:

Check Groups		Check Names
Document Checks		Dimensioning Standard, Arrow Style, Note Font, Dimension Font, Detail Font, Section Font, View Arrow Font, Table Font, Balloon Font
Drawing Document Checks		Sheet Format, Layer, Title Block

In SolidWorks Design Checker, click **File** and select **Create Checks from DWG**.

See **Check Documents from DWG Files** in the help.

Criticality Level Checks

You can set the importance level of checks based on **Criticality**. The settings include: **Critical**, **High**, **Medium**, and **Low**.

To view criticality levels:

1. Open any **.swstd** file and under **Criticality**, set the levels, as needed, for each check.
2. Check the document, and under **Design Checker**, expand **Failed Checks**.

   All **Failed Checks** are now organized by **Criticality**.

Font Checks Duplication

You can duplicate the values of one font check to other font checks across all tabs that include font checks without re-entering any values.

To duplicate font checks in a new standards file:

1. Click **Build Checks** and select **Create a New Standards File**.
2. On the **Document Checks** tab, select any font check, such as **Note Font** .
3. In the **Create Font Check** dialog box:
   a) Under **Font Parameters**, select the appropriate values.
   b) Under **Copy To**, select the appropriate font types.
4. Click **Apply** to display a panel for each font type you selected.
5. Click **OK** in the **Create Font Check** dialog box, and continue creating the new standards file.

To duplicate font checks on existing **.swstd** files, select **Open an Existing Standards File** and select any font check not previously included in the file to display the **Create Font Check** dialog box.

Multiple Standard Files on Active Document

You can concurrently select multiple **.swstd** files when checking active documents. Previously, checking documents based on multiple **.swstd** files required separate check sequences for each **.swstd** file.

To use multiple standard files:

1. With an active document, click **Check Active Document** to display the **Select Standards** dialog box.
2. Click **Add**, browse for **.swstd** files, and select all the standard files you want to include.
3. Click **Open** to add the files.
4. If required, select files and click **Remove** or **Move Up** and **Move Down**.
5. Click **Validate** to check the active document and display the results.

An alert appears if the same check is found in multiple standard files.

Document Checks

Custom Property

The operator **!=** (**Not Equal To**) was added to the **Operator** list.

If	you specify **Not Equal To** as the **Operator**
and	the property **Name** and **Value** are identical to those specified under the **Custom** tab in **Summary Information**
then	the check fails.

Dimension Checks

Replaced Original Text

Cycles through all dimensions in a SolidWorks document to verify that the original dimension text was not modified.

Text Position

Adds text position checks.

To assign checks for text position:

1. Select **Text Position** .
2. Click one value from each of the four dimension text groups:

Linear Dimension Text

Radial Dimension Text

Angular Dimension Text

Chamfer Dimension Text

Dimension Precision

Verifies the number of digits after decimal points for dimension and for tolerance values. Checks are run for the **Primary Dimension** and the **Angular Dimension.**

To assign checks for dimension precision:

1. Select **Dimension Precision** x.xx .
2. Select precision values for each set:

Primary Dimension	Angular Dimension
Select the number of digits after the decimal point for dimension values.	
Select the number of digits after the decimal point for tolerance values.	

Drawing Document Checks

Display Mode

Cycles through drawing documents to verify that the **Drawing Views** are using the expected **Display Mode**.

To assign checks for display mode:

1. Select **Display Mode** .
2. Select the display modes to add.

Part Document Checks

Bend Tables

Select **Bend Table** to verify that sheet metal documents are using the correct bend tables. When you check the document, the check passes if the list contains the specified bend tables.

To assign checks for bend tables:

1. Click ... to browse, and select the **Bend Table Files** (***.xls**) to include.
2. Click **Open** to add the files.

Assembly Document Checks

Suppressed Components

Cycles through all parts and configurations in assembly documents (top level assembly only) for any suppressed part.

Feature Checks

Suppressed Features

Cycles through all features and model configurations for suppressed parts.

For **Suppressed Components** , and **Suppressed Features** :

- The check fails only if the condition exists in all configurations.
- Setting the **Criticality** level is the only user interface.

SolidWorks Task Scheduler

Task Scheduler supports new task types.

Export PDMWorks Files

You can use Task Scheduler to perform batch exports of PDMWorks Workgroup files.

Options include the following:

- Including subprojects
- Specifying target file types (***.dxf**, ***.dwg**, ***.igs**, ***.pdf**, ***.step**, ***.jpg**).
- Filtering files in a project based on type
- Exporting to a folder you specify, to the PDMWorks vault, or to both.
- Exporting as PDF

To export PDMWorks Workgroup files:

1. Click **Export PDMWorks Workgroup Files** .
2. Log in with your PDMWorks Workgroup user name and password and the name of the computer with the PDMWorks Workgroup vault.
3. In the **Export PDMWorks Files** dialog box, set options, then click **Finish**.

Render and Animation

You can use Task Scheduler to manage rendering and animation tasks you set up in PhotoWorks or Motion Studies.

To monitor scheduled tasks:

Select **Renderings and Animations**.

Dissect Files

You can use Task Scheduler to set the frequency and timing of file dissection for Design Clipart.

The SolidWorks files you search are now dissected to make their components easy to locate and reuse.

When you dissect files:

- Parts are dissected into features (extrudes and cuts).
- Features are dissected into sketches.
- Drawings are dissected into general tables and blocks.
- DWG/DXF files are dissected into tables, blocks, and views.

Thumbnail images created during dissection are displayed as thumbnails when you do a SolidWorks search.

See **Design Clipart** on page 1-10.

SolidWorks Utilities

Compare Documents - Assemblies

Additional properties are supported when you compare assembly documents:

- Unique parts
- Unique sub-assemblies
- Total resolved components
- Total fully resolved components
- Total suppressed features
- Total fully lightweight components
- Total bodies

Feature Paint

Feature Paint supports sheet metal features except bend, corner-trim, fold, unfold, and welded corner.

Find Replace Annotations

You can find annotations in these tables:

- BOM
- General table
- Hole table
- Revision table
- Weldment cut list

Simplify Enhancements

- Revolves are supported (volume-based only).
- You can simplify multiple feature types (for example fillets and extrudes). Previously you could simplify only one feature type at a time.
- When **Create derived configurations** is cleared, you can add the simplified features to a different configuration you select under **Configurations**. You can also rename a configuration here, and it updates in the FeatureManager design tree. **Configurations** lists only the active configuration and its derived configurations.

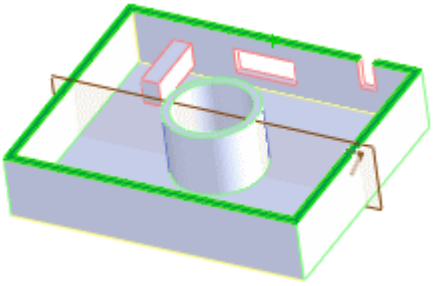
Symmetry Check

Symmetry Check checks for geometric symmetry in parts about a plane. It identifies symmetrical, unsymmetrical, and unique faces. You can generate and save a results report.

In parts with multiple configurations, symmetry check works with the current configuration only. Once you identify symmetrical entities, you can work on a portion of the model, which saves you design time and improves performance.

To check for symmetry in parts:

1. Click **Symmetry Check** (Utilities toolbar) or **Utilities**, **Symmetry Check**.
2. Define the plane of symmetry and select appropriate entities in the graphics area to create the plane.
3. Click **Check** to view the results.



Example - unsymmetrical face

4. Click to close the PropertyManager or **Recheck** to run another symmetry check.

See **Symmetry Check** in the help.

Thickness Analysis

Two options let you specify the **Thickness scale** display under **Color Settings**:

Continuous
Uses a gradient range

Discrete
Uses a discrete number of color bands

PDMWorks Workgroup

Portable Document Format (PDF) Support

To make it easier to exchange documents, you can:

- Automatically create PDF in the vault during check in
- Generate PDF from the Task Scheduler
- Preview any PDF document in the vault
- View document property changes with eDrawings

Creating PDF Files During Check-In

A PDF option has been added to the PDMWorks VaultAdmin to enable the automatic creation of a PDF file during the document check-in process. When this option is enabled, each time a SolidWorks drawing is checked in, a PDF file with the same name is associated with it.

These PDF files are not attachments and are not visible anywhere in the user interface to prevent modification of the filename in the vault.

When the PDF file is generated, a new command on the shortcut menu lets you open the file in Acrobat Reader. This command is also available from the Web Portal and SolidWorks Explorer.

When you modify a drawing and check it back in, the PDF file is updated automatically.

Creating PDF Files Using the Task Scheduler

The SolidWorks Task Scheduler allows you to schedule a task to export PDMWorks Workgroup files in Adobe Portable Document Format (.pdf), as well as other export file types supported by SolidWorks. A PDF file is created for each drawing in the projects being exported.

You can also use the Task Scheduler to update all PDF files in batch mode.

When you generate PDF files using the Task Scheduler, you can specify whether the files are stored outside the vault or inside the vault.

See **Export PDMWorks Files** on page 13-20.

Preview PDF Documents in the Vault

You can preview any PDF file from:

- SolidWorks with the PDMWorks Add-in
- SolidWorks Explorer
- PDMWorks Web Portal

You can open PDF files that are associated with drawings automatically with Acrobat Reader if it is already installed. If Acrobat Reader is not installed, you are prompted to install it.

Local and vault previews for JPG, BMP, GIF, PNG, TXT, XML, HTM and HTML files are also supported in SolidWorks Explorer.

Viewing Changed Document Properties with eDrawings

Changes you make to document properties from PDMWorks Workgroup, SolidWorks Explorer, and Windows Explorer can be viewed in eDrawings without opening the files in SolidWorks. This includes.

- Bumping a document's revision

  When the document is a drawing, you can view the changes that have been made to the fields in the title block.

- Custom Properties

For methods to modify drawing properties from SolidWorks Explorer and Windows Explorer, see **Modifying Document Properties** on page 1-10.

PDMWorks Workgroup Vault Import and Export

Two utilities are provided to export PDMWorks Workgroup projects from one vault and import them into another vault. This makes it possible to have the same information in vaults at two different sites, or to merge two vaults into one.

Users with administrator privileges can export project documents and their metadata to an XML file that can then be imported into another PDMWorks Workgroup vault. In addition to documents, administrators, users, and groups that are defined in the project can be exported. Administrators can export all projects from a vault or select specific projects to export.

Toolbox

Auto Size Toolbox Components

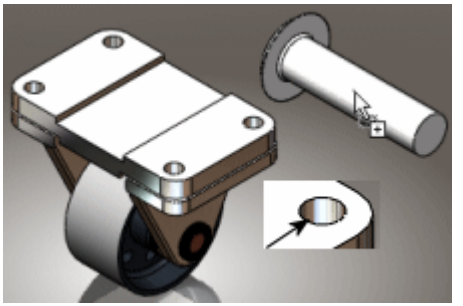
Selected toolbox components include auto sizing capability. With auto sizing, Toolbox components adapt to the size of the geometry to which they are dragged.

Auto Sizing Toolbox Components

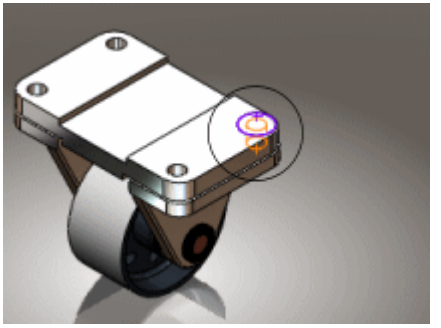
- Bolts and screws
- Nuts
- Retaining rings
- Pins
- Washers
- Bearings
- O-Rings
- Gears

To use auto resizing Toolbox components:

1. From the Toolbox folder, select a component and drag it to graphics area.
2. Select the hole where you want to place the component.



The component resizes and an exact preview is displayed.

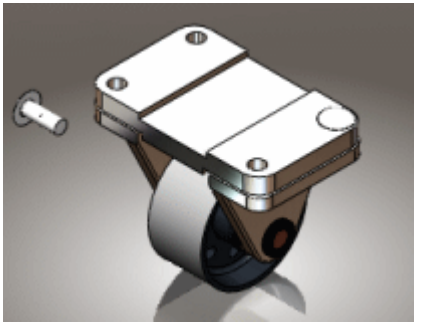


3. In the PropertyManager:
   a) Under **Properties**, adjust the values.

b) Under **Options**, select **Auto size to mated geometry**.

When you select **Auto size to mated geometry**, the component updates when you change the geometry to which the component is mated.

4 Drag the next component, that includes your adjustments, and place it where needed.



5 When done, click ✔.

Content

Toolbox has added content in the following standards for these types:

Standards	Descriptions
ANSI Inch	Flat countersunk, Undercut flat countersunk, and Hex washer head tapping screws, etc.
ANSI Metric	Pan head and Hex head tapping screws, etc.
ISO	Hexalobular socket cheese head screws, and pan head screws, etc.
BSI	External and internal retaining rings, Keys (one end and two end square and rectangular), etc.
MIL	100° and 82°, flat head, cross-recessed; Hex head cap, Hex socket, Hex plain, etc.

See **SolidWorks Toolbox Overview** in the help.

For other Toolbox-related functionality, see:

- **Smart Fasteners** on page 6-21.
- **Hole Series** on page 4-4.

14

SolidWorks Office Premium Components

This chapter describes enhancements to the SolidWorks Office Premium components in the following areas:

- ScanTo3D
- SolidWorks Routing
- TolAnalyst

ScanTo3D

General

Merging Meshes and Removing Overlaps

ScanTo3D can merge meshes and trim overlaps to produce a valid individual mesh.

To automatically merge meshes:

1. In the **Open** dialog box, select a mesh file containing multiple meshes.
2. Click **Options**.
3. Select **Merge meshes to remove overlaps** and click **OK**.
4. Click **Open**.

To manually merge meshes or sub-meshes:

1. Open a file containing multiple meshes.
2. Select the meshes or sub-meshes to merge, right-click, and select **Merge Meshes**.

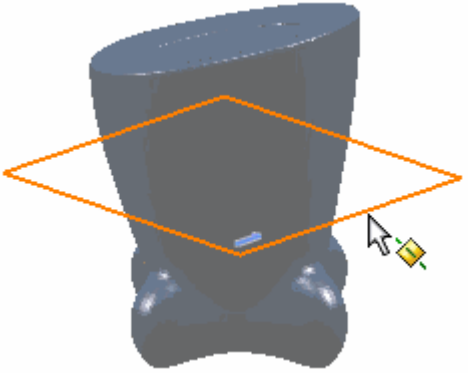
   The selected meshes are listed in the PropertyManager.
3. Click ✔.

Sketching on Mesh Intersecting with Section Plane

For mesh files, you can sketch along the intersection of the section view plane and the mesh, referencing the pierce point where the plane intersects a facet edge. The sketch is retained when you close the section view. This functionality lets you create reference sketches that were previously not possible.

To sketch on a mesh using the section view plane:

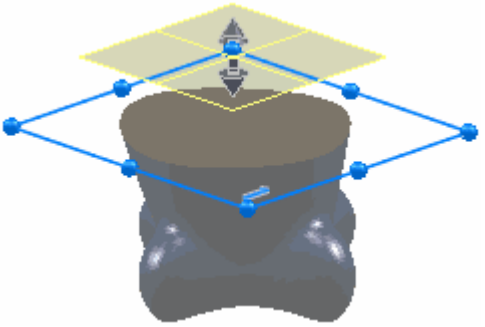
1. Create and position the plane for the section view, then select it.



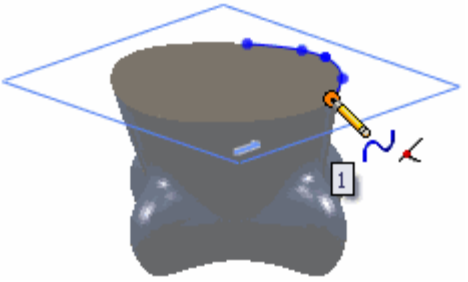
2 Click **Section View** (View toolbar) or **View**, **Display**, **Section View**.

3 Click .

4 Select the plane and open a 2D sketch.

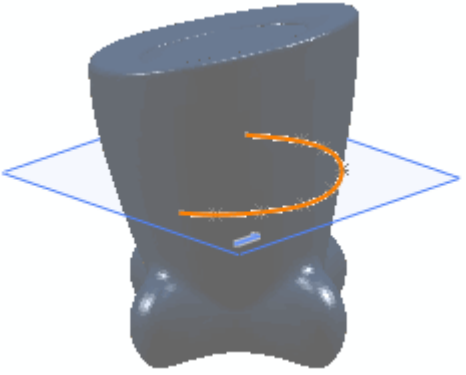


5 Use the **Spline** tool (Sketch toolbar) to sketch points on the section view plane that reference the mesh.



As you sketch and reference the mesh, note that the sketch now inferences the pierce point where the plane intersects a facet edge. This functionality produces a more accurate sketch than was previously possible.

6 Exit the sketch and click to clear **Section View** .



Textures

ScanTo3D automatically imports textures from 3D Studio **.3ds**, **.obj**, and **.wrl** files if the textures are clearly linked to the mesh file. Otherwise, the mesh imports with no textures. You can specify the location of the texture files.

To specify the location to save the texture files.

1. Click **Open** (Standard toolbar) or **File**, **Open**.
2. In the **Open** dialog box, select **Mesh Files** in **Files of type**.
3. Browse to a mesh file in **File name**.
4. Click **Options**.
5. In the **Import Options** dialog box, for **Texture files directory**, click **Browse**.
6. Browse to the folder, then click **OK** twice.
7. Click **Open**.

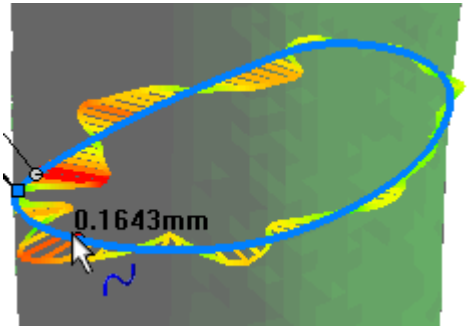
Deviation Analysis

The Deviation Analysis tool displays deviations for these types of analyses:

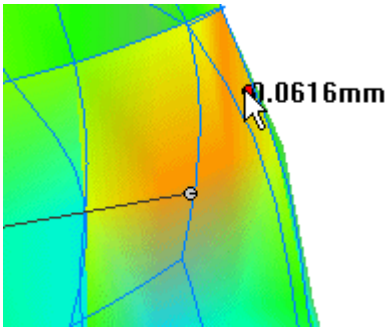
- **Curve to Mesh**. Colored spines along the curve show the deviation between the curve and the mesh.
- **Surface to Mesh**. A colored display on the mesh shows the deviation between the extracted surface and the mesh.
- **Mesh to Mesh**. Make an original mesh to use as a reference to create another working mesh. Measure back to the original mesh to ensure the working mesh remains within tolerance.

Hover over the deviation for specific deviation values. You can save a detailed deviation report.

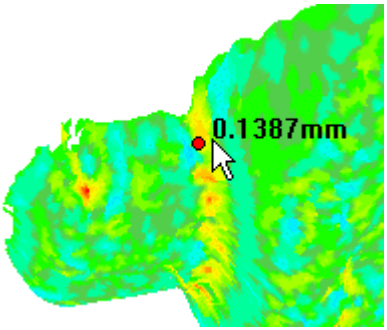
Curve to mesh deviation



Surface to mesh deviation



Mesh to mesh deviation



To measure mesh deviation:

1. Open a file with curves or surfaces referencing a mesh.
2. Click **Deviation Analysis** (ScanTo3D toolbar) or **Tools**, **ScanTo3D**, **Deviation Analysis**.
3. Select an **Analysis Type**.
4. Under **Analysis Parameters**, select the mesh and curves or surfaces.
5. Adjust the **Sensitivity** slider and upper and lower boundaries, then click **Calculate**.

   Colored spines or patches show the deviation. A callout indicates the maximum, minimum, average, and standard deviations.
6. Hover over the deviation colors for exact local measurements.
7. Click .

See **Deviation Analysis PropertyManager** in the help.

Mesh Editing Tools

You can move, copy, scale, or offset meshes. Click **Mesh Edit** (ScanTo3D toolbar) or **Tools**, **ScanTo3D**, **Mesh Edit**. Set the options in the **Mesh Edit** PropertyManager.

See **Mesh Edit PropertyManager** in the help.

Curve Wizard

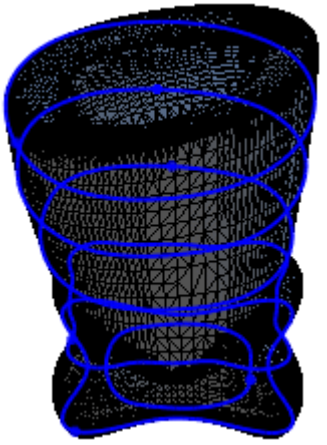
The new Curve Wizard can create boundary and section curves in point cloud or mesh files. A boundary forms along boundary edges. ScanTo3D can create boundary curves only along boundary edges.

To use the Curve Wizard:

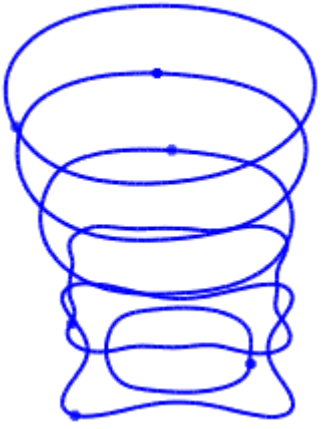
1. Open a point cloud or mesh file.
2. Click **Curve Wizard** (ScanTo3D toolbar) or **Tools**, **ScanTo3D**, **Curve Wizard**.
3. Select the mesh or point cloud.
4. In the PropertyManager, under **Creation Method**, select a curve type:
   - **Section Curve**
   - **Boundary Curve**
5. Set the remaining PropertyManager options.

   For section curves, under **Section Plane Parameters**, you can specify these entities for **Section Plane**:
   - A pair of points.
   - A plane.
   - An existing curve, so you can create section curves perpendicular to it.
6. Click .

   The curves are created as a 3D sketch.



Section curves on a mesh



Section curves with mesh hidden

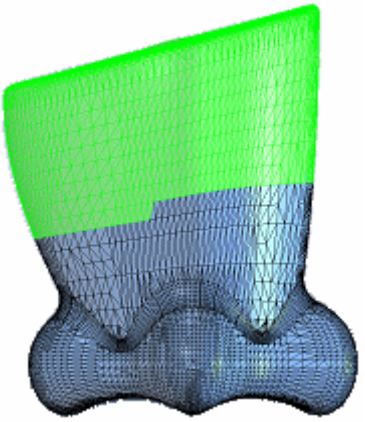
See **Curve Wizard PropertyManager** in the help.

Mesh Prep Wizard

Boundary Smoothing

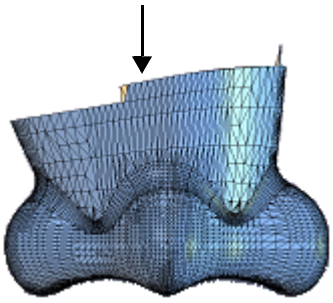
Several options allow you to smooth mesh boundaries in the Mesh Prep Wizard.

- In the **Extraneous Data Removal** PropertyManager: **Trim mesh boundary to selection**. Trim a mesh boundary exactly to the selection boundary, smoothing all serrated boundaries.

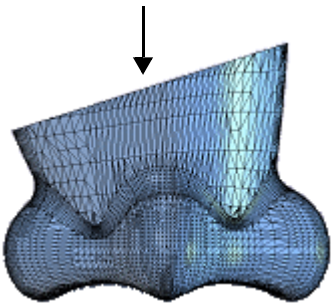


Area box-selected for removal

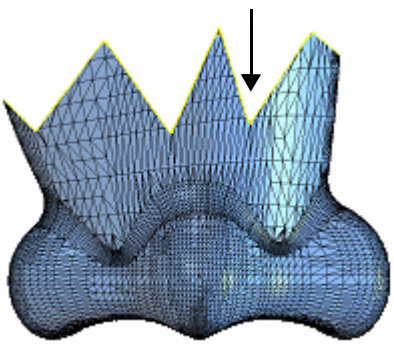
Option cleared =
Serrated boundary



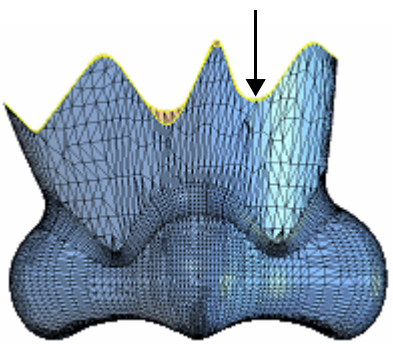
Option selected =
Smooth boundary



- In the **Smoothing** PropertyManager. Move the **Boundary Smoothness** slider to smooth serrated boundaries.



No **Boundary Smoothness**



Boundary Smoothness applied

Simplification PropertyManager

Local Simplification

Use tools under **Local Simplification** to simplify selected areas of the mesh. Use local simplification in conjunction with the **Invert selection** option to preserve specific areas of design intent while simplifying the rest of the model. **Global Simplification** still exists.

Thin Parts

The **Optimize thin mesh** option improves recognition of point cloud data as a thin mesh.

Point cloud

Mesh optimized for thin mesh

See **Simplification PropertyManager** in the help.

Surface Wizard

General Enhancements

- Surface Wizard creates surfaces with fewer patches, improving performance and usability.
- Surface Wizard creates overbuilt surfaces, minimizing the amount of work you need to do to extend them prior to trimming operations. The surfaces also perform better when you have to extend them.

Feature Lines

Surfaces created by ScanTo3D are composed of patches (ideally four-sided). Groups of patches form regions on the surfaces. Feature lines form the boundaries between regions.

Ideally, feature lines are located where the mesh curvature transitions between a contact through tangent relationship. For example, at a filleted edge separating two sides of a cube.

You can edit feature lines in the **Automatic Surface Creation** PropertyManager so you can extract more desirable surfaces. You can create and delete lines plus move lines and vertices.

See **Automatic Surface Creation PropertyManager** in the help.

Hands-on Example

B-Spline Surfaces

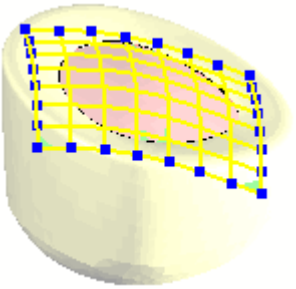
When extracting b-spline surfaces, a mesh lets you visualize the surface. You can add curves from the mesh into the U and V directions for the extracted surface, forcing the surface to fit more closely to the mesh.

You can display a deviation analysis to show the deviation between the mesh and the surface.

To use the mesh to extract b-spline surfaces:

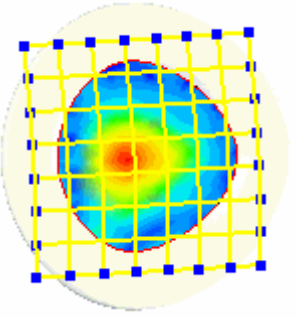
1. Open a mesh file, run the Surface Wizard, and using guided creation, advance to the **Surface Extraction** PropertyManager.
2. Select a surface, then under **Face Settings**, click **Bspline** .

   A surface with a mesh appears.



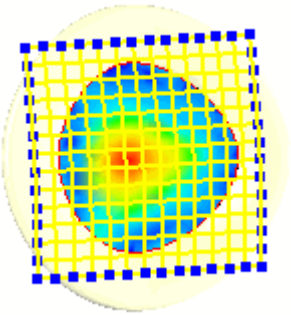
3. Rotate the model to look directly at the surface.
4. Under **Face Settings**, select **Display deviation analysis**.

   The deviation between the surface and the mesh is displayed. A color range legend appears in the graphics area. Surface deviation is around the red, yellow, and green-colored area.



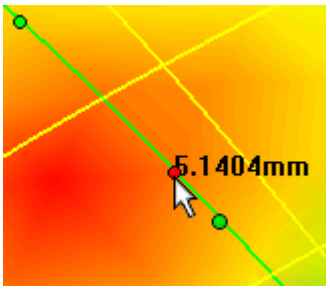
5 To improve surface accuracy:

- Use the **U-Direction** and **V-Direction Number of Segments** in the PropertyManager to add U and V curves.

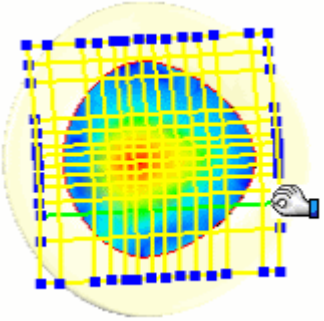


Add U and V curves *before* editing the position of curves. Modifying the number of curves automatically repositions all curves.

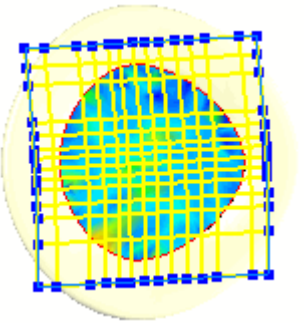
- You can click in two different locations directly on the mesh to create U and V curves.



- Click **Editing UV curves**, select a curve from the list, then drag to reposition it. Repeat as necessary.



- Increase the number of U and V lines over the area with the highest deviation to try to reduce the deviation.



6 Click **Rebuild Surface** to incorporate the edits into the surface and review the new deviation analysis.

See **Deviation Analysis** on page 14-5.

Surface Extraction Types

In the **Surface Extraction** PropertyManager, you can extract these specific surface types listed under **Face Settings**:

Surface type	Icon	Parameters to Specify
Extrude		Specify a single wall of adjacent mesh faces that form a surface, similar to an extrude.
Revolve		Specify a single wall of adjacent mesh regions that form a surface, similar to a revolve.
Ruled		Specify a single mesh region to form a surface, similar to a two-profile loft with no guide curves.
Torus		Specify a single mesh region to form a surface, similar to a face created by a fillet.

For **Extrude** and **Revolve**: All mesh faces must be part of one mesh to be extracted together as an extrude or revolve. Use the Surface Wizard's paint tools in the **Face Identification** PropertyManager to identify the mesh faces with the same color.

See **Surface Extraction PropertyManager** in the help.

SolidWorks Routing

General

Enhancements to the workflow and usability in SolidWorks Routing:

- You enter the name of new route sub-assemblies and parts in the **Route Properties** PropertyManager, instead of separate dialog boxes.
- A route sub-assembly is not created until you click ✔ in the **Route Properties** PropertyManager. If you cancel out of the PropertyManager, the sub-assembly is not created. Previously, the sub-assembly was created before the PropertyManager was launched, so it was created even if you cancelled the command.
- The **Route Properties** PropertyManager displays only those fields pertinent to the type of route you are creating and the method you are using to create it.
- When you start a route by right-clicking an end connector and selecting **Start Route**, you can right-click anywhere on the connector. Previously, you had to click on the connection point.
- You can access the **Fixed Length** PropertyManager by right-clicking a route segment (tube or electrical cable) and selecting **Fixed Length**.
- When you right-click a route segment, **Route Segment Properties** has replaced **Route Properties** on the shortcut menu.
- When creating a route on the fly, you can end the route without a routing component.
- When automatically routing through clips, you can flip which side of the clip the cable enters.
- After placing a component in a route, you can undo the placement without having to exit the route sketch.

Electrical Connectors with Multiple Connection Points

The software now automatically starts a route when you drop a connector with multiple connection points into an assembly.

Fixed Length

If you apply **Repair Route** while the **Fixed Length** option is on, the software resolves the violation of the minimum bend radius. A warning appears if the repair requires the length to be changed. The resulting length value is the new fixed length.

Exception: If the cable is a standard cable, the fixed length always stays the same.

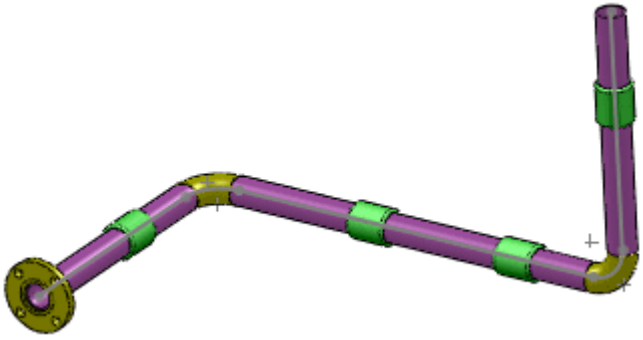
Lightweight Connection Points and Route Points

The software no longer needs to resolve a lightweight component to access data contained in its connection points and route points.

Stock Lengths for Pipes

When creating a new pipe route assembly, you can specify a standard length of pipe to use. Straight sections in the route are divided into standard lengths, with the last section being created as a fraction of the specified standard length. You can:

- Change the standard length or remove it.
- Use different standard lengths in the same assembly, applying to different straight segments.
- Automatically insert couplings between the segments.



Options

Routing options have been reorganized, and are grouped by:

- General Routing Settings
- Piping and Tubing
- Electrical Cabling

You can set routing file locations for:

- Routing Library
- Routing Template
- Standard Tubes
- Cable/Wire Library
- Component Library
- Standard Cables

The file settings automatically propagate to the appropriate fields in the electrical library wizards, the **Routing Components Wizard**, and the **Route Properties** PropertyManager. Additionally, to facilitate sharing the above settings with other users, you can save the settings to a file or load previously saved settings.

If you change the **Routing Library** folder location, the software uses the new folder location when opening existing route assemblies and their routing components.

See **Routing Options** and **Routing File Locations** in the help.

Auto Route

The **Auto Route** PropertyManager and functionality have been enhanced:

- **Automatic Repair of Route**. If, upon route creation, there is a minimum bend radius violation, the software automatically repairs the route instead of prompting you to fix it.
- **Undo button**. The **Undo** button has been moved to the standard location at the top of the PropertyManager.
- **Merging Guidelines**. When merging guidelines, you can:
  - Select clips to route through at the same time.
  - Bias the resulting route segment toward one of the end connectors.
  - Choose to merge on one end only instead of both ends.

  Additionally:
  - When you join guidelines to existing routes, a split point is automatically created if one does not already exist where you select to join.
  - To make selection easier, you can filter guidelines by wire or cable name and signal name, based on the names imported by the from-to list.

Flattened Route

You can flatten an electrical route sub-assembly within the 3D model. The flattened route includes cable and wires, coverings, and end connectors. You can specify to display the connectors as 3D connectors or connector drawing blocks. The flattened route is a simplified representation of the route. The lines are not true lengths, but the dimensions on the route segments display the true length of the routed wire.

The flattened route is created as a new configuration of the 3D electrical route sub-assembly. In the FeatureManager design tree, a **FlattenedRoute** feature is added. To toggle between the 3D and flattened route, right-click **Route** or **FlattenedRoute** and select **Show Configuration**.

In the flattened route:

- You can change the presentation by dragging entities and constraining them. These changes are not applied to the 3D configuration.
- You cannot make geometric changes (such as adding or deleting branches). You make these changes in the 3D configuration, and the flattened configuration updates.

To flatten a route, in the route sub-assembly, click **Flatten Route** (Electrical toolbar) or **Routing**, **Electrical**, **Flatten Route**.

See **Flatten Route** in the help.

TolAnalyst

TolAnalyst is a tolerance analysis application that determines the effects that dimensions and tolerances have on parts and assemblies. The TolAnalyst tools let you perform “worst-case” tolerance stack-up analysis on parts and assemblies.

First you use the DimXpert tools to apply dimensions and tolerances to the parts or components in an assembly, then you use the TolAnalyst tools to leverage that data for stack-up analysis.

TolAnalyst is available only in SolidWorks Office Premium.

See **DimXpert for Parts** on page 10-3.

To ensure you have valid tolerance data, TolAnalyst uses a wizard interface that walks you through a four-step procedure:

1. Create a measurement between two features.
2. Create an assembly sequence between the measurement features.
3. Apply constraints to each part.
4. Evaluate the results.

Hands-on Example

Index

3D
  blocks 3-3
  instant3D 3-6
  symmetry and mirror 3-2
  textures, ScanTo3D 14-4
64-bit, COSMOSWorks 11-2

A

add-ins, activating 1-3
Adobe
  Illustrator 12-13
  Photoshop 12-13
alignment
  dimensions 9-2
  holes 6-6
  sketch entities 9-4
Analysis Advisor, COSMOSWorks 11-2
annotations
  SolidWorks Utilities 13-22
  views in drawings 9-4
API 12-2–12-6
appearances 2-3–2-6
  applying 2-3
  editing 2-6
  emissive appearances 13-10
assemblies 6-1–6-21
  display states 6-2, 6-18
  filtering 6-14
  hole alignment 6-6
  large 6-2, 6-17
  layouts 6-7
  motors 8-4
  performance 6-2, 6-17
  Pro/ENGINEER 12-14
  references 6-2
  selection in 6-14
  simplified representations 6-17
  Smart Fasteners 6-21
  statistics 6-2
  virtual components 6-7
AssemblyXpert 6-2
auto sizing, SolidWorks Toolbox 13-26
auto trace, sketching 3-5
Autodesk Inventor 12-13

B

balloons 9-2
beams, COSMOSWorks 11-12
bend tables, SolidWorks Design Checker 13-19
bills of materials 9-5
bodies, isolating in parts 5-4
bolts, COSMOSWorks 11-14
bonding, COSMOSWorks 11-16
boundary curves, ScanTo3D 14-7
boundary smoothing, ScanTo3D 14-8
boundary surfaces
  linear option 4-2
  tangent influence 4-2
broken views 9-3
b-spline surfaces, ScanTo3D 14-11
bushing properties 6-11

C

capturing parameters, DriveWorksXpress 12-9
circular patterns 4-8
color properties, RealView 2-3
Color/Image tab 2-3
columns in tables 9-8
CommandManager 1-3
compare documents, SolidWorks Utilities 13-22
components
  configurations 7-2
  copying mates with 6-12
  derived 6-3
  patterns 6-3
  selecting 6-14
  selective loading 6-18
  show hidden 6-16
  virtual 6-7

configurations 7-1–7-7
assemblies 7-6
creating 7-4
design tables 7-2
modifying 7-4
parts 7-4
PropertyManager 7-2
constraints
DimXpert 10-8
Pro/ENGINEER 12-14
contact, COSMOSWorks 11-16
context toolbars 1-6
copy
drawing sheets 9-3
mates 6-12
CornerXpert 4-3
cosmetic patterns 4-8
COSMOSMotion 8-4–8-5
dampers 8-4
fixed and floating parts 8-5
legacy studies 8-5
mates 8-4
plots 8-5
redundant constraints 8-5
COSMOSWorks 11-1–11-20
64-bit 11-2
Analysis Advisor 11-2
beams 11-12
bolt connectors 11-14
contact 11-16
damping 11-4
design insight plots 11-19
design scenarios 11-13
dynamics 11-3
harmonic analysis 11-5
large displacement 11-13
linear dynamic studies 11-3
load path plots 11-19
mesher 11-15
modal time history analysis 11-4
Mooney-Rivlin material models 11-2
nitinol 11-2
nonlinear dynamic studies 11-8
Ogden material models 11-2
pin connectors 11-15
pressure vessel studies 11-3
radiation view factors 11-11
random vibration 11-7
shells 11-3
strains in nonlinear studies 11-18
stress linearization 11-20
structural members 11-12
time history graphs 11-18
trend tracker 11-13
units 11-2
coupler joints 6-10
curvature constraints 3-9
Curve Wizard, ScanTo3D 14-7
custom properties
DriveWorksXpress 12-9
parts 5-2
SolidWorks Design Checker 13-18
SolidWorks Explorer with PDMWorks Workgroup 1-10
SolidWorks File Explorer 1-10
Windows Explorer 1-10
customizing the CommandManager 1-3

D

damping, COSMOSWorks 11-4
datums, DimXpert 10-5
delete tolerances, DimXpert 10-8
derived components 6-3
derived parts, breaking links 5-2
design
COSMOSWorks insight plots 11-19
COSMOSWorks scenarios 11-13
top-down 6-7
Design Clipart 1-10
design tables 7-2
detailing 9-1–9-8
deviation analysis, ScanTo3D 14-5
DFMXpress
configuring parameters 12-8
design rules 12-7
dimensioning and tolerancing 10-1–10-11
dimensions
alignment 9-2
DimXpert 10-5, 10-7
modify 7-2
properties 9-2
SolidWorks Design Checker 13-19
TolAnalyst 14-17
DimXpert for parts 10-3–10-10
constraints 10-8
datums 10-5
delete tolerances 10-8
dimensions 10-5, 10-9
DimXpertManager 10-4
features 10-3
geometric tolerances 10-5
geometric tolerancing 10-6
location dimension 10-7
model feature recognition 10-4
options 10-4
overview 10-3
patterns 10-5
size dimension 10-5
TolAnalyst 14-17
tolerances 10-9
topology recognition 10-4
displacement, COSMOSWorks 11-13
display states 6-2, 6-18, 12-12
dissect files
Design Clipart 1-10
SolidWorks Task Scheduler 13-21

document properties, modifying 1-10
document selection
  browsers 1-8
  preview tooltips 1-9
drawing views 9-3–9-4
drawings 9-3–9-4
  copying sheets 9-3
  from part/assembly documents 9-3
  in eDrawings 12-11
  inserting images 9-3
DriveWorksXpress 12-8
DXF/DWG 12-13
dynamics 11-3–11-11
  damping 11-4
  example of linear dynamic study 11-8
  harmonic analysis 11-5
  linear dynamic studies 11-3
  modal time history analysis 11-4
  nonlinear dynamic studies 11-8
  random vibration 11-7

E

edge flanges, FeatureWorks 13-2
eDrawings 12-11–12-12
embedded components 6-7
emissive appearances 13-10
equations 9-6
export 12-13–12-14
exporting flat patterns 12-13, 12-16

F

feature lines, ScanTo3D 14-10
Feature Paint, SolidWorks Utilities 13-22
FeatureManager design tree
  filtering 1-4, 6-14
  show/hide items 1-4
features 4-1–4-10
  boundary surfaces 4-2
  circular patterns 4-8
  CornerXpert 4-3
  cosmetic patterns 4-8
  fillets 4-3
  Hole Series 4-4
  Hole Wizard 4-5
  Instant3D 4-6
  recognition, DimXpert 10-4
  split lines 4-9
  split parts 4-9
  sweeps 4-10
FeatureWorks 13-2–13-4
  base sweeps 13-4
  edge flanges 13-2
  hems 13-3
  resize tool 13-3
file export, PDMWorks files 13-20
fillets 4-3
filtering 6-14
Firefox 12-11
fitting text 9-7
flattened routes 14-16
flyout menus 1-9
fonts
  in tables 9-7
  SolidWorks Design Checker 13-17
friction properties 6-11

G

gamma correction, PhotoWorks 13-13
GD&T 10-2, 14-17
geometric tolerancing 14-17
geometric tolerancing, DimXpert 10-5, 10-6
graphics cards, diagnosing in SolidWorks Rx 12-17

H

hands-on examples
  Adding a Motor 8-4
  DFMXpress 12-8
  DriveWorksXpress 12-8
  Layout-based Design 6-9
  Using Mate References to Position Inserted Parts 5-3
  Using RealView 2-10
  Using TolAnalyst 14-17
  Welded Corner 12-16
  Working with Feature Lines 14-10
harmonic analysis, COSMOSWorks 11-5
Heads-up View toolbar 1-6
hems, FeatureWorks 13-3
hide/show in tables 9-7
hole alignment 6-6
hole depth/diameter ratio, DFMXpress 12-7
Hole Series 4-4, 6-21
Hole Wizard 4-5, 7-2

I

Illustrator 12-13
images in drawings 9-3
import 12-13–12-14
inaccessible features, DFMXpress 12-7
in-context components 6-7
indirect illumination 13-7
input forms, DriveWorksXpress 12-9
installation 12-2
Instant3D 4-6
Inventor 12-13

J

joints
  coupler 6-10
  screw 6-11
  universal 6-11

L

layouts 6-7
leaders 9-2
legacy studies, COSMOSMotion 8-5
linear coupler mates 6-10
linear dynamics, COSMOSWorks 11-3
linear option, boundary surfaces 4-2
links, breaking 5-2
load bearing faces 6-11
load references 6-11
location dimension, DimXpert 10-7
lock mates 6-10

M

manufacturability, DFMXpress 12-7
manufacturing features 10-3
mapping 2-3–2-6
  cubic environment, PhotoWorks 13-6
  DXF/DWG 12-13
  size 2-6
  style 2-3
materials. *See* appearances
mates 6-9–6-14
  bushings 6-11
  coordinate system 6-9
  copying 6-12
  COSMOSMotion 8-4
  friction 6-11
  icons 6-9
  linear/linear coupler 6-10
  load bearing faces 6-11
  lock 6-10
  origin 6-9
  path 6-10
  references 6-12
  references in parts 5-3
  screw 6-11
  universal joint 6-11
memory management, PhotoWorks 13-13
Menu Bar 1-2
mesher, COSMOSWorks 11-15
meshes, ScanTo3D 14-2, 14-5, 14-6
message display, controlling 1-10
mirrored parts, breaking links 5-2
misaligned holes 6-6
modal time history, COSMOSWorks 11-4
MoldflowXpress 12-16
molds 12-16
Mooney-Rivlin, COSMOSWorks 11-2
motion studies 8-1–8-5
MotionManager 8-3
motors 8-4
Mozilla Firefox 12-11

N

nitinol, COSMOSWorks 11-2
nonlinear dynamics, COSMOSWorks 11-8

O

Ogden, COSMOSWorks 11-2
Open Document browser 1-8
options
  PhotoWorks 13-7
  routing 14-15

P

parts 5-1
  breaking links 5-2
  custom properties 5-2
  isolating bodies 5-4
  positioning when inserting 5-3
  sketches 5-2
  solid bodies 5-2
path mates 6-10
patterns
  circular 4-8
  component 6-3
  cosmetic 4-8
  DimXpert 10-5
PDMWorks Workgroup 13-24–13-25
  bump revision 13-25
  document properties 13-25
  exporting vault projects 13-25
  file export 13-20
  importing vault projects 13-25
performance 6-2
Photoshop 12-13
PhotoWorks 13-5–13-15
  batch document processing 13-5
  cubic environment mapping 13-6
  emissive appearances 13-10
  gamma correction 13-13
  image adjustment 13-14
  indirect illumination 13-7, 13-13
  materials and scenes 13-9
  materials PropertyManager 13-9–13-12
  realistic falloff 13-15
  scene editor 13-11
  system options 13-13
physical simulation 8-4
pins, COSMOSWorks 11-15
pipes, stock length 14-15
point clouds, Mesh Prep Wizard 14-9
Portable Document Format (PDF)
  creating during check-in 13-24
  creating using the Task Scheduler 13-24
  previewing in PDMworks Workgroup 13-25
  support in PDMWorks Workgroup 13-24
pressure vessels, COSMOSWorks 11-3

preview
eDrawings 12-11
PDMWorks Workgroup PDF files 13-25
SolidWorks files 13-21
prismatic parts, DFMXpress 12-8
Pro/ENGINEER 12-11, 12-14
PropertyManager, sketching 3-4

R

random vibration, COSMOSWorks 11-7
RealView 2-1–2-14
appearances 2-3
legacy models 2-2
scenes 2-8
using 2-2
workflow 2-2
Recent Document browser 1-8
rectangles 3-4
references 6-2
remove mesh overlap, ScanTo3D 14-2
revisions, bumping 13-25
Rhino files 12-14
Routing. *See* SolidWorks Routing
RSS feed, eDrawings 12-11
ruled surfaces, ScanTo3D 14-13
rules in DriveWorksXpress 12-9

S

save animation, MotionManager 8-3
scales, SolidWorks Utilities 13-23
ScanTo3D 14-2–14-13
boundary curves 14-7
boundary smoothing 14-8
b-spline surfaces 14-11
Curve Wizard 14-7
deviation analysis 14-5
feature and patch boundaries 14-10
feature lines 14-10
local simplification 14-9
merge meshes 14-2
mesh editing tools 14-6
remove mesh overlap 14-2
section curves 14-7
simplifying locally 14-9
sketch on mesh 14-2
surface extraction types 14-13
surface wizard 14-10
textures 14-4
thin parts 14-9
trim mesh boundary 14-8
scenes 2-8–2-13
applying 2-10
basic 2-8
editing 2-11–2-13
PhotoWorks editor 13-11
presentation 2-9
studio 2-10
screw mates 6-11
search 1-2
section curves, ScanTo3D 14-7
section views 9-4
selecting components 6-14
sheet metal
export flat pattern 12-13, 12-16
weld beads 12-16
shells, COSMOSWorks 11-3
shortcut toolbars 1-7
show hidden components 6-16
show/hide sketch 3-6
simplify, SolidWorks Utilities 13-22
simplifying locally, ScanTo3D 14-9
size dimension, DimXpert 10-5
sketch entity alignment 9-4
sketching 3-1–3-10
3D symmetry and mirror 3-2
area/hatch fill 3-3
auto trace 3-5
blocks 3-3
instant 3D 3-6
parts 5-2
ScanTo3D mesh 14-2
show/hide 3-6
splines 3-8–3-10
SketchXpert 3-7
Smart Fasteners 6-21
solid bodies 5-2
solid sweeps 4-10
SolidWorks Design Checker 13-16–13-20
criticality levels 13-16
custom properties 13-18
dimension checks 13-18–13-19
drawing documents 13-19
DWG files 13-16
font duplications 13-17
multiple standards files 13-17
parts documents 13-19
SolidWorks Explorer
preview documents 12-15
tags 12-15
SolidWorks Office Premium 14-1
SolidWorks Office Professional 13-1
SolidWorks Routing 14-14–14-17
auto route 14-16
connection points 14-14, 14-15
file locations 14-15
fixed length 14-14
flattened routes 14-16
options 14-15
route properties 14-14
stock length for pipes 14-15
SolidWorks Rx 12-17
SolidWorks Search 1-2
SolidWorks Task Scheduler 13-20–13-22

SolidWorks Toolbox
  auto size components 13-26
  content 13-27
  Smart Fasteners 6-21
SolidWorks Utilities 13-22–13-23
  annotations 13-22
  compare documents, assemblies 13-22
  Feature Paint 13-22
  simplify 13-22
  symmetry check 13-23
  thickness analysis 13-23
splines 3-8–3-10
  continuity at handles 3-8
  curvature constraints 3-9
  spline on surfaces 3-10
split lines 4-9
split parts 4-9
standards files, SolidWorks Design Checker 13-17
STL files 12-12
stress, COSMOSWorks 11-20
sub-assemblies, selecting 6-14
suppressed components, SolidWorks Design Checker 13-20
surfaces
  ScanTo3D 14-13
  spline on surfaces 3-10
sweeps 4-10, 13-4
symmetry, SolidWorks Utilities 13-23

T

tables 9-6–9-8
tags
  documents and features 1-6
  SolidWorks Explorer 12-15
tangent influence, boundary surfaces 4-2
Task Pane 1-9
Task Scheduler
  dissecting files 13-21
  PDMWorks file export 13-20
  renderings and animations 13-21
text
  notes 9-7
  SolidWorks Design Checker 13-18
  tables 9-7
textures, ScanTo3D 14-4
thickness, SolidWorks Utilities 13-23
thin parts, ScanTo3D 14-9
thumbnails for SolidWorks files 13-21
time history, COSMOSWorks 11-18
TolAnalyst 10-11, 14-17
tolerance analysis 14-17
tolerances
  DFMXpress 12-7
  DimXpert 10-9
toolbars
  context-sensitive 1-6
  Heads-up View 1-6
  Menu Bar 1-2
  shortcut 1-7
top-down design 6-7
topology recognition, DimXpert 10-4
torus surfaces, ScanTo3D 14-13
trends, COSMOSWorks 11-13
trim mesh boundary, ScanTo3D 14-8
turned parts, DFMXpress 12-8

U

undo dimension properties 9-2
units, COSMOSWorks 11-2
universal joint mates 6-11
updates, installation 12-2
UV curves, ScanTo3D 14-11

V

video capture, Solidworks Rx 12-17
views. *See* drawing views
virtual components 6-7

W

weld beads 12-16
weldments 9-5, 12-18

X

XPS (XML Paper Specification) 12-12, 12-14