



# **SolidWorks® 99**

## **User's Guide**

© 1999, SolidWorks Corporation  
Concord, Massachusetts 01742  
All Rights Reserved.

U.S. Patent 5,815,154

SolidWorks Corporation is a Dassault Systemes S.A. (Nasdaq:DASTY) company.

Information is subject to change without notice. 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 Corporation.

As a condition to your use of this software product, you agree to accept the limited warranty, disclaimer and other terms and conditions set forth in the SolidWorks Corporation License and Subscription Service Agreement which accompanies this software. If, after reading the License Agreement, you do not agree with the limited warranty, the disclaimer or any of the other terms and conditions, promptly return the unused software and all accompanying documentation to SolidWorks Corporation and your money will be refunded.

SolidWorks® is a registered trademark of SolidWorks Corporation.

SolidWorks® 99 is a product name of SolidWorks Corporation.

FeatureManager™, Feature Palette™, and PhotoWorks™ are trademarks of SolidWorks Corporation.

ACIS® is a registered trademark of Spatial Technology Inc.

IGES™ Access Library is a trademark of IGES Data Analysis, Inc.

FeatureWorks™ is a trademark of Geometric Software Services Co. Limited.

Other brand or product names are trademarks or registered trademarks of their respective holders. All warranties given by SolidWorks Corporation 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.

Document Number: SWXUGENG061599

The information and the software discussed in this document are subject to change without notice and should not be considered commitments by SolidWorks Corporation.

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.

#### 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 252.227-7013(c)(1)(ii)(Rights in Technical Data and Computer Software) and in this Agreement, as applicable.

Contractor/Manufacturer:  
SolidWorks Corporation, 300 Baker Avenue,  
Concord, Massachusetts 01742.

Portions of this software are copyrighted by and are the property of Unigraphics Solutions Inc.

Portions of this software © 1995 - 1999  
D-Cubed Limited.

Portions of this software © 1992-1999  
Summit Software Company.

Portions of this software © 1990-1999  
LightWork Design Limited.

Portions of this software © 1995-1999  
Spatial Technology Inc.

Portions of this software © 1998-1999  
Geometric Software Services Co. Limited.

Portions of this software © 1999  
Immersive Design, Inc.

The IGES Access Library portion of this product is based on IDA IGES Access Library © 1989-1998 IGES Data Analysis, Inc.  
All Rights Reserved.

# Contents

---

<b>Chapter 1</b>	<b>SolidWorks Fundamentals</b>	<b>1-1</b>
Installing the SolidWorks 99 Software . . . . .		1-2
System Requirements . . . . .		1-2
Required Information for Installation . . . . .		1-2
Installing the SolidWorks 99 Software . . . . .		1-4
SolidWorks Service Packs . . . . .		1-5
Converting Older SolidWorks Files to SolidWorks 99 . . . . .		1-5
Visiting the SolidWorks Web Site . . . . .		1-5
Basic Concepts . . . . .		1-6
Getting Help . . . . .		1-8
Opening a SolidWorks Part, Drawing, or Assembly . . . . .		1-9
Opening a New SolidWorks Document . . . . .		1-9
Opening an Existing SolidWorks Document . . . . .		1-9
Opening an Existing Part from Explorer . . . . .		1-10
SolidWorks Terms . . . . .		1-12
Document Windows . . . . .		1-14
Toolbars . . . . .		1-16
Customize and Arrange Toolbars . . . . .		1-17
The FeatureManager Design Tree . . . . .		1-18
Symbols and Conventions . . . . .		1-19
FeatureManager Design Tree Options . . . . .		1-20
What's Wrong? . . . . .		1-20
Rebuild Symbol . . . . .		1-21
Drag and Drop Features . . . . .		1-21

Shortcut Menus . . . . .	1-22
Selection Methods . . . . .	1-23
Selecting from the Graphics Area . . . . .	1-23
Selecting Hidden or Coincident Items . . . . .	1-25
Selecting from the FeatureManager Design Tree . . . . .	1-25
Viewing and Manipulating Parts, Assemblies, and Drawings . . . . .	1-26
The View Toolbar . . . . .	1-26
The Standard Views Toolbar . . . . .	1-28
Using Named Views . . . . .	1-29
Multiple Views . . . . .	1-30
Time Saving Tips . . . . .	1-31
Keyboard Shortcuts . . . . .	1-32
Preparation for Printing . . . . .	1-33
Page Setup . . . . .	1-33
Object Linking and Embedding . . . . .	1-35
Linking vs. Embedding . . . . .	1-35
Using Data from Other Applications . . . . .	1-36
Bringing SolidWorks Data into Other Applications . . . . .	1-37
2D Sketching and Command Emulator . . . . .	1-38
Rendering Models with the PhotoWorks Application . . . . .	1-38
Identifying Features with FeatureWorks . . . . .	1-38
SolidWorks Application Programming Interface . . . . .	1-39

**Chapter 2 Sketching 2-1**

Starting a New 2D Sketch . . . . .	2-2
The Sketch Toolbar . . . . .	2-3
The Sketch Relations Toolbar . . . . .	2-4
The Sketch Tools Toolbar . . . . .	2-5
Using the Sketch Entity Tools . . . . .	2-7
Using the Sketch Tools . . . . .	2-14
Working in a Sketch . . . . .	2-26
The Sketch Origin . . . . .	2-27
The Sketch Grid . . . . .	2-27
Inferencing Pointers and Lines . . . . .	2-27
Automatic Inferencing Lines . . . . .	2-29
Using Silhouette Edges . . . . .	2-29
The Shortcut Menu . . . . .	2-30

Cutting, Copying, and Pasting Multiple Sketch Entities . . . . .	2-30
Copying and Pasting Entire Sketches . . . . .	2-30
Creating a Sketch on the Face of a Part . . . . .	2-31
Editing a Sketch . . . . .	2-31
Exiting a Sketch . . . . .	2-32
Edit Sketch Planes. . . . .	2-32
Derived Sketch . . . . .	2-32
Adding Dimensions and Relations. . . . .	2-33
Dimensioning Sketches. . . . .	2-33
Geometric Relations . . . . .	2-37
Add Relations Table . . . . .	2-39
Display/Delete Relations . . . . .	2-41
Fully Defining Sketches . . . . .	2-44
PropertyManager Functionality . . . . .	2-45
Does it matter where I start sketching? . . . . .	2-48
How complex should sketches be?. . . . .	2-49
3D Sketching . . . . .	2-50

**Chapter 3 Reference Geometry 3-1**

Creating a Construction Plane . . . . .	3-2
Moving, Resizing, and Copying Planes . . . . .	3-4
Changing Plane Names. . . . .	3-5
Hide or Show Planes. . . . .	3-5
Changing the Plane of a Sketch . . . . .	3-5
Creating an Axis. . . . .	3-6
Creating a Coordinate System . . . . .	3-7
Converting Sketch Lines to Construction Geometry . . . . .	3-8
Curve . . . . .	3-9
Projected Curve. . . . .	3-9
Composite Curve . . . . .	3-11
Helix . . . . .	3-11
Split Line. . . . .	3-12
Curve Through Reference Points . . . . .	3-14
Curve Through Free Points. . . . .	3-15
Displaying Reference Geometry . . . . .	3-16

<b>Chapter 4</b>	<b>Creating Features</b>	<b>4-1</b>
The Features Toolbar		4-2
Base/Boss		4-2
Cut		4-2
Extrude		4-2
Specifying End Conditions		4-4
Thin Features		4-6
Extruded Base/Boss		4-7
Extruded Cut		4-8
Revolve		4-9
Sweep		4-10
Simple Sweep		4-10
Sweep with Guide Curves		4-11
Start and End Tangency Options for Sweeps		4-14
Loft		4-15
Simple Loft		4-15
Loft with Non-planar Profiles		4-16
Loft with Split Line		4-17
Loft with Guide Curves		4-18
Center Line Loft		4-20
Start and End Tangency Options for Lofts		4-21
Fillet/Round		4-23
Constant Radius		4-23
Variable Radius		4-24
Face Blend		4-25
Face Blend with Hold Line		4-26
Overflow Type		4-27
Chamfer		4-28
Draft		4-29
Neutral Plane		4-29
Parting Line		4-30
Hole		4-32
Scale		4-33
Shell		4-34
Rib		4-35
Dome		4-36
Shape		4-37

Pattern and Mirror . . . . .	4-40
Linear Pattern . . . . .	4-41
Circular Pattern . . . . .	4-42
Mirror Feature . . . . .	4-44
Mirror Pattern . . . . .	4-44
Mirror All . . . . .	4-45
Pattern of Patterns . . . . .	4-45
Controlling and Modifying Patterns . . . . .	4-46
Surfaces . . . . .	4-50
Creating Surfaces . . . . .	4-50
Knit Surfaces. . . . .	4-54
Using Surfaces to Create Features . . . . .	4-56

**Chapter 5 Working with Parts 5-1**

Feature Editing . . . . .	5-2
Edit Definition . . . . .	5-2
Part Color and Appearance . . . . .	5-2
Properties of Features and Faces. . . . .	5-4
Parent/Child Relationships . . . . .	5-5
Moving and Resizing Features . . . . .	5-5
Copying Features . . . . .	5-7
Controlling Access to Dimensions . . . . .	5-8
Equations . . . . .	5-9
Measuring Sizes and Distances . . . . .	5-11
Dependency Editing . . . . .	5-13
Suppressing Features . . . . .	5-13
Rollback . . . . .	5-15
Derived Parts . . . . .	5-16
Base Part . . . . .	5-16
Mirror Part . . . . .	5-16
Derive Component Part. . . . .	5-17
Editing Derived Parts . . . . .	5-17
File Management with External References. . . . .	5-18
Checking Model Geometry . . . . .	5-19
Force Regeneration with Verification on Rebuild . . . . .	5-19
Check Entity . . . . .	5-20
Custom Properties . . . . .	5-21
Displaying Annotations in Parts. . . . .	5-22
Section View of a Model . . . . .	5-23

Mass Properties and Section Properties .....	5-25
Lighting .....	5-27
Light Types .....	5-27
Light Properties .....	5-28
Curvature .....	5-31
Sending Part Documents .....	5-32

**Chapter 6 Working with Assemblies 6-1**

Design Methods .....	6-2
Bottom-Up Design .....	6-2
Top-down Design .....	6-2
Understanding Assemblies .....	6-3
The Assembly Window .....	6-3
The Assembly Toolbar .....	6-4
The FeatureManager Design Tree in an Assembly .....	6-5
Creating an Assembly .....	6-6
Adding Components to an Assembly .....	6-6
Deleting a Component from an Assembly .....	6-8
Opening Assemblies .....	6-9
Managing the Part Files of an Assembly .....	6-10
Saving an Assembly and Its Parts .....	6-10
Verifying Which Components Are Used in an Assembly .....	6-10
Backing Up an Assembly .....	6-11
Replacing a Component in an Assembly .....	6-12
Positioning Components in an Assembly .....	6-17
Assembly Mating .....	6-19
Mating Relationships .....	6-19
Mategroups .....	6-19
Creating a Mating Relationship .....	6-20
Modifying a Mating Relationship .....	6-22
Smart Mates .....	6-22
Deleting a Mating Relationship .....	6-27
Suppressing a Mating Relationship .....	6-27
Errors in Solving Mates .....	6-27
Working with Sub-assemblies .....	6-28
Creating a Sub-assembly .....	6-28
Modifying a Sub-assembly .....	6-30
Editing a Sub-assembly in Context .....	6-33



Exploding an Assembly View .....	6-35
Using AutoExplode .....	6-35
Creating an Exploded View .....	6-35
Exploding and Collapsing an Exploded View .....	6-37
Editing an Exploded View .....	6-37
Detecting Interference Between Components .....	6-38
Simplifying Large Assemblies .....	6-39
Toggling the Visibility .....	6-39
Suppression States .....	6-40
Lightweight Parts .....	6-40
Changing the Suppression State .....	6-43
Assembly Envelopes .....	6-45
Component Selection by Properties .....	6-47
Advanced Component Selection .....	6-47
Advanced Show/Hide Components .....	6-48
Saving and Reloading Selection Criteria .....	6-49
Customizing the Appearance of an Assembly .....	6-49
Displaying Annotations in an Assembly .....	6-51
Assembly Statistics .....	6-51
Sending Assembly Documents .....	6-52
<b>Chapter 7 Working with Parts within an Assembly</b> .....	<b>7-1</b>
External References .....	7-2
Setting External Reference Options .....	7-3
Searching for Referenced Documents .....	7-4
Saving Assemblies with In-context Features .....	7-5
Editing Parts in the Assembly Context .....	7-6
Creating a Part in an Assembly .....	7-6
Editing a Part in an Assembly .....	7-7
Opening Documents from Part or Assembly Windows .....	7-8
Time-Dependent Features .....	7-9
Adding a Component Pattern .....	7-11
Reorder and Roll Back .....	7-13
Using an Assembly Layout Sketch .....	7-14
Working with Assembly Features .....	7-16
Specifying the Scope .....	7-16
Creating an Assembly Feature .....	7-16

Creating Molds . . . . .	7-18
Simple Mold. . . . .	7-18
Mold with Complex Parting Lines. . . . .	7-21
Core and Cavity Mold . . . . .	7-22
Joining Parts . . . . .	7-24
Welding . . . . .	7-26
Surface Delta and Radius. . . . .	7-26
General Procedure for Welding Components . . . . .	7-27
Face Selection Tips . . . . .	7-28
Weld Symbols . . . . .	7-28

**Chapter 8 Working with Configurations 8-1**

Creating a Configuration Manually. . . . .	8-2
Configurable Items. . . . .	8-3
Activating a Configuration . . . . .	8-4
Editing a Configuration. . . . .	8-5
Opening a Document in a Named Configuration . . . . .	8-5
Special Assembly Configurations . . . . .	8-6
Component Configurations in an Assembly . . . . .	8-7
Design Tables . . . . .	8-8
Creating a Design Table. . . . .	8-8
Specifying Design Table Parameters . . . . .	8-12
Summary of Design Table Parameters. . . . .	8-17
Design Tables in Drawings . . . . .	8-18

**Chapter 9 Drawings 9-1**

Drawing and Page Setup Options . . . . .	9-2
Creating a Drawing . . . . .	9-3
Modifying the Sheet Setup. . . . .	9-3
Multiple Sheets . . . . .	9-4
Templates, Sheets, and Views . . . . .	9-5
Customizing a Drawing Template. . . . .	9-5
The Drawing Window . . . . .	9-9
Views of Parts and Assemblies . . . . .	9-10
View Boundaries . . . . .	9-11
Standard 3 View . . . . .	9-12
Projection View . . . . .	9-14
Auxiliary View . . . . .	9-15

Named View . . . . .	9-16
Relative to Model View . . . . .	9-17
Detail View . . . . .	9-18
Broken View . . . . .	9-20
Section View . . . . .	9-21
Aligned Section View . . . . .	9-22
Modifying Section Views . . . . .	9-23
Section Views of Assemblies . . . . .	9-26
Troubleshooting Section Views . . . . .	9-27
View Properties . . . . .	9-28
Updating Views . . . . .	9-29
View Alignment and Display . . . . .	9-30
Moving Views . . . . .	9-30
Aligning Views . . . . .	9-31
Rotating Views . . . . .	9-31
Copying and Pasting Views . . . . .	9-32
Hiding and Showing Views and Edges . . . . .	9-32
Display Mode . . . . .	9-33
Tangent Edge Display . . . . .	9-34
Line Formats and Layers . . . . .	9-35
Component Line Font . . . . .	9-35
Line Format . . . . .	9-35
Layers . . . . .	9-36
Printing a Drawing . . . . .	9-37
Sending Drawing Documents . . . . .	9-38
2D Sketching . . . . .	9-39
Empty Drawing View . . . . .	9-39
Importing Sketch Geometry into a Drawing . . . . .	9-39
2D Emulator . . . . .	9-39
<b>Chapter 10 Detailing</b> . . . . .	<b>10-1</b>
Detailing Options . . . . .	10-2
Annotating Drawing Documents . . . . .	10-2
Dimensions in a Drawing . . . . .	10-3
Importing Model Dimensions . . . . .	10-4
Adding Reference Dimensions . . . . .	10-5
Baseline Dimensions . . . . .	10-6
Ordinate Dimensions . . . . .	10-7
Moving and Copying Dimensions . . . . .	10-8

Aligning Dimensions . . . . .	10-9
Modifying Witness Lines . . . . .	10-11
Modifying Dimension Values . . . . .	10-12
Dimension Properties . . . . .	10-12
Dimension Visibility . . . . .	10-17
Reference Geometry in a Drawing . . . . .	10-17
Annotations . . . . .	10-18
Importing Model Annotations . . . . .	10-18
Creating Annotations . . . . .	10-19
Aligning Annotations . . . . .	10-19
Note . . . . .	10-20
Center Mark . . . . .	10-24
Hole Callout . . . . .	10-24
Cosmetic Thread . . . . .	10-25
Surface Finish Symbol . . . . .	10-26
Datum Feature Symbol . . . . .	10-27
Datum Target . . . . .	10-28
Geometric Tolerancing Symbol . . . . .	10-29
Bill of Materials (BOM) . . . . .	10-31
Balloon Callout . . . . .	10-38
Weld Symbol . . . . .	10-39
Area Hatch . . . . .	10-40
Block . . . . .	10-41
Design Table . . . . .	10-42

## **Chapter 11 Library Features and the Feature Palette Window 11-1**

Library Features . . . . .	11-2
Creating a Library Feature . . . . .	11-2
Adding a Library Feature to a Part . . . . .	11-3
Editing a Library Feature . . . . .	11-4
Adding Color . . . . .	11-5
Dissolving a Library Feature . . . . .	11-5
The Feature Palette Window . . . . .	11-6
Displaying the Feature Palette Window . . . . .	11-7
Organizing Feature Palette Items . . . . .	11-8
Creating a Palette Item . . . . .	11-9
Working with Palette Items and Folders . . . . .	11-11
Adding Palette Items to Documents . . . . .	11-12
Palette Parts as Derived Parts . . . . .	11-14

<b>Chapter 12 Sheet Metal</b>	<b>12-1</b>
Designing Sheet Metal Parts . . . . .	12-2
Bend Types . . . . .	12-3
Sheet Metal Features in the FeatureManager Design Tree . . . . .	12-4
Bend Allowance . . . . .	12-7
Auto Reliefs . . . . .	12-9
Creating a Sheet Metal Part . . . . .	12-10
Using Sharp Bends . . . . .	12-10
Using Filleted Corners . . . . .	12-12
Using Cylindrical or Conical Faces . . . . .	12-13
Using Rips . . . . .	12-14
Using a Flat Model . . . . .	12-16
Adding Features in the Flattened State . . . . .	12-17
Adding Bends to a Flattened Model . . . . .	12-18
Hems . . . . .	12-18
Bends Across Multiple Tabs . . . . .	12-19
Editing Bends . . . . .	12-20
Editing Bend Parameters . . . . .	12-20
Editing Auto Reliefs . . . . .	12-22
Creating a Flat Pattern Configuration . . . . .	12-22
Creating Drawings of Sheet Metal Parts . . . . .	12-23
Tips for Designing Sheet Metal Parts . . . . .	12-24
Forming Tools . . . . .	12-24
Creating Forming Tools . . . . .	12-24
Creating Positioning Sketches . . . . .	12-26
Controlling Access to Dimensions . . . . .	12-27
Saving and Adding Forming Tools to the Feature Palette . . . . .	12-27
Applying Forming Tools . . . . .	12-27
<b>Chapter 13 Piping</b>	<b>13-1</b>
Overview . . . . .	13-2
General Procedure . . . . .	13-2
3D Sketch . . . . .	13-3
Required Features in Piping Parts . . . . .	13-3
Connection Points . . . . .	13-4
Connection Point Parameters in the Design Table . . . . .	13-5
Route Points . . . . .	13-6

Routing Components . . . . .	13-7
Pipe Parts . . . . .	13-7
Elbow Parts . . . . .	13-8
Flange Parts . . . . .	13-10
Reducers . . . . .	13-10
Other Fittings . . . . .	13-11
Routing Options . . . . .	13-12
Creating a Route . . . . .	13-14
Editing a Route . . . . .	13-16
Editing the 3D Sketch . . . . .	13-16
Adding Fittings Manually . . . . .	13-17
Pipe to Pipe Penetration . . . . .	13-18
Reorganizing a Routing Sub-assembly . . . . .	13-19
Drawings of Pipe Assemblies . . . . .	13-20

**Chapter 14 Importing and Exporting Files 14-1**

Importing/Exporting SolidWorks Documents . . . . .	14-2
Importing Documents . . . . .	14-2
Setting Import Options . . . . .	14-3
Editing Imported Features . . . . .	14-4
Import Diagnosis . . . . .	14-5
Exporting Documents . . . . .	14-6
IGES Files (.igs, .iges) . . . . .	14-7
Exporting IGES Files . . . . .	14-8
Importing IGES Files . . . . .	14-9
Parasolid Files (.x_t, .x_b) . . . . .	14-10
STEP Files (.step) . . . . .	14-11
ACIS Files (.sat) . . . . .	14-11
VDAFS Files (.vda) . . . . .	14-11
STL Files (.stl) . . . . .	14-12
DXF/DWG Files (.dxf, .dwg) . . . . .	14-13
Open DXF/DWG File . . . . .	14-13
Attach Dimensions . . . . .	14-13
Sketch from Drawing . . . . .	14-13
DXF/DWG Export Options . . . . .	14-14
DXF/DWG Mapping . . . . .	14-14
VRML Files (.wrl) . . . . .	14-17
TIFF Files (.tif) . . . . .	14-17

FeatureWorks . . . . .	14-19
FeatureWorks Options . . . . .	14-20
Feature Recognition . . . . .	14-20
<b>Appendix A SolidWorks 99 Options</b>	<b>A-1</b>
Color Options . . . . .	A-2
Crosshatch Options . . . . .	A-3
Detailing Options . . . . .	A-4
Drawings Options . . . . .	A-8
Edges Options . . . . .	A-10
External References Options . . . . .	A-11
General Options . . . . .	A-13
Grid/Units Options . . . . .	A-16
Line Font Options . . . . .	A-17
Material Properties Options . . . . .	A-18
Performance Options . . . . .	A-18
Reference Geometry Options . . . . .	A-20
Routing Options . . . . .	A-21
<b>Appendix B SolidWorks Application Programming Interface</b>	<b>B-1</b>
Recording and Playing SolidWorks Macros . . . . .	B-2
Using Visual Basic to Develop an Application . . . . .	B-3
Programmer's Guide . . . . .	B-4
<b>Appendix C SolidWorks Corporation License and     Subscription Service Agreement</b>	<b>C-1</b>

# List of Tables

---

Table 5-1.	Functions available in equations. . . . .	5-11
Table 6-1.	Geometry types supported in mating relationships . . . . .	6-19
Table 6-2.	Alignment conditions in mating relationships . . . . .	6-21
Table 6-3.	Smart mates . . . . .	6-23
Table 6-4.	Comparison of component suppression states . . . . .	6-44
Table 7-1.	Weld types . . . . .	7-26
Table 8-1.	Summary of design table parameters . . . . .	8-17
Table 14-1.	Import and Export File Types. . . . .	14-2
Table 14-2.	Interactive Feature Recognition . . . . .	14-23



# SolidWorks Fundamentals

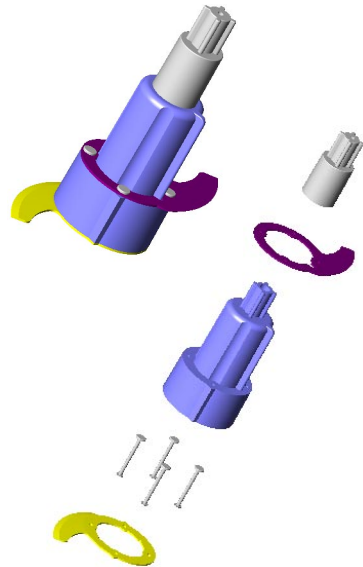
---

## ***What is SolidWorks 99?***

SolidWorks® 99 is mechanical design automation software that takes advantage of the familiar Microsoft® Windows® graphical user interface. This easy-to-learn tool makes it possible for mechanical designers to quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

This chapter discusses some basic concepts and terminology used throughout the SolidWorks 99 application. It provides an overview of the following topics:

- Installing the SolidWorks 99 software
- SolidWorks terms and document windows
- FeatureManager™ design tree
- Finding out what is wrong
- Shortcut menus and time saving tips
- Selecting items, and displaying parts, assemblies and drawings
- Object Linking and Embedding (OLE)
- Rendering models with PhotoWorks™
- Feature recognition with FeatureWorks™
- 2D sketching and command line emulator
- Using the SolidWorks API development environment



## Installing the SolidWorks 99 Software

---

### System Requirements

- Microsoft Windows NT 4.0 (with Service Pack 4), Windows 95, or Windows 98.  
Some SolidWorks 99 functionality is limited when using Windows 95. For up-to-date information, refer to the SolidWorks 99 Release Notes.
- Pentium®-based computer, or an Alpha-based computer
- 64 MB RAM minimum recommended
- Mouse or other pointing device
- CD-ROM drive
- If you plan to use the Design Table or Bill of Materials features in SolidWorks, you are strongly recommended to install Microsoft Office 97 Service Release 2 (SR-2). Installing SR-2 fixes a Microsoft problem that causes either, or both, Excel and SolidWorks processes to remain running when you exit SolidWorks.
- To open SolidWorks files with Internet Explorer, you must be using Internet Explorer, version 4.0 or later.

---

**NOTE:** You should make backup copies of all SolidWorks documents (parts, assemblies, and drawings) created with a previous version of the software before opening them in SolidWorks 99.

---

### Required Information for Installation

Before you install SolidWorks 99, if you made changes to palette items, drawing templates, or PhotoWorks materials in a previous release of the SolidWorks software, you should save copies of these files in a backup folder so they will not be lost or overwritten during the installation.

The SolidWorks 99 CD installation program guides you through the installation procedure and asks for the following information:

- The type of installation:
  - **Individual.** This installation is the most common choice. It should be selected for a computer that will not share its SolidWorks installation with any other computers, whether or not it is on a network.
  - **Client/Server.** This installation should be selected for a computer that will run SolidWorks, and will also act as a server, sharing its SolidWorks installation with one or more other computers.

---

**NOTE: The server and clients must be of the same type platform. There is no cross-platform installation.**

---

- **Server Only.** This installation should be selected for a computer that will *not* run the SolidWorks 99 application, but will only act as a server, sharing its SolidWorks installation with one or more other computers (SolidWorks clients). (You must reinstall if you want to change this selection.) With this option, the server does not need to be licensed to run the SolidWorks application, but each client must be licensed.

---

**NOTE:** Before starting a SolidWorks update installation, it is important to ensure that *no* SolidWorks clients are running.

---

- ❑ **SolidWorks Serial Number.** For new SolidWorks customers or non-subscription customers, the serial number is located on the back of the SolidWorks CD-ROM case. Subscription customers should continue to use the serial number on the CD-ROM case from their first SolidWorks installation.
- ❑ **SolidWorks Registration Code:**
  - If you are a SolidWorks subscription customer, the registration wizard provides a registration code during the SolidWorks 99 installation.
  - If you are a new SolidWorks customer, you are asked to use the registration wizard during installation. This generates a registration form that you can email or fax to SolidWorks Corporation. A registration code will be provided quickly by return email or fax. You have up to 30 days to use the product without the registration code.
  - If you are a continuing, non-subscription customer, you must have a registration code to update to the SolidWorks 99 software.
- ❑ **PhotoWorks Serial Number.** If you have just purchased the PhotoWorks photo-realistic rendering application, you need to type the PhotoWorks serial number, located on the back of the CD-ROM case. If you are a SolidWorks subscription customer continuing to use PhotoWorks, use your original PhotoWorks serial number.
- ❑ **FeatureWorks Serial Number.** If you have just purchased FeatureWorks feature recognition software, you need to type the FeatureWorks serial number, located on the back of the CD-ROM case. If you are a SolidWorks subscription customer continuing to use FeatureWorks, use your original FeatureWorks serial number.
- ❑ **Enable or disable MCD.** This option applies only to Windows NT 4.0. You should enable MCD (Mini-Client Driver) if you are using a graphics card that has Mini-Client Driver support so that SolidWorks can take advantage of the card's ability to accelerate 3D OpenGL.
- ❑ **Allow or disallow model changes from drawing.** This option makes it possible for a user to disallow the ability to make changes to part or assembly dimensions from the drawing. (You must reinstall the SolidWorks software to change this option.) The default, and most common configuration, is to allow the part and assembly dimensions to be changed from the drawing.

## Installing the SolidWorks 99 Software

---

The installation steps for an individual user, a server, or a client/server installation are the same. The answers to the questions asked during the installation may differ. For example, a serial number is not required for a computer that is used exclusively as a server.

### **To install the SolidWorks software:**

- 1 Insert the CD-ROM in the drive.
- 2 Within a few seconds the install wizard will begin the installation and guide you through the few questions that you have to answer.

### **The Client/Server Installation**

The computer designated as the file server shares its SolidWorks installation with one or more other computers. However, before each of the clients can run the SolidWorks application, it is necessary to perform a SolidWorks client installation to prepare the client computer for sharing the application. The actual SolidWorks program files will not be installed on the client, but some setup procedures must take place. See the client installation discussed below.

The server and clients must be the same platform type; there is no cross-platform installation.

---


**NOTE:** A Serial Number and Registration Code are required for each SolidWorks Client and Client/Server.

---

### **To perform a SolidWorks client installation:**

- 1 Working on the SolidWorks server computer, make sure that the folder containing the SolidWorks installation is shared so that it is accessible to the client computers on the network.
- 2 Working on the SolidWorks client computer, browse to the folder under the SolidWorks installation directory called:  
On Alpha Windows NT: **setup\alpha**  
On Intel Windows NT, Windows 95, or Windows 98: **setup\i386**
- 3 Double-click on the file named **setup.exe**.
- 4 The installation guides you through the few steps needed to complete the installation.

---

**NOTE:** After installation, to learn about the new functionality in SolidWorks 99, click  or **Help, SolidWorks 99 Help Topics** on the main toolbar.

---

## SolidWorks Service Packs

If you are a SolidWorks subscription customer, you can take advantage of SolidWorks service packs that are regularly posted on the SolidWorks Web site. These service packs contain software updates and enhancements to the SolidWorks 99 software. To check for a new service pack, click **Help, Service Packs**, and click the **Check** button. Click the check box if you want the software to automatically check the SolidWorks Web site for a new service pack once a week.

## Converting Older SolidWorks Files to SolidWorks 99

---

Because of changes to the SolidWorks files with the development of SolidWorks 99, opening a SolidWorks document from an earlier release may take more time than you are used to experiencing. However, once the file has been opened and saved, subsequent opening time will return to normal.

The SolidWorks Conversion Wizard provides a way for you to automatically convert all of your SolidWorks files from an earlier version to the SolidWorks 99 format. Depending on how many files you have, the conversion process may take a while, but once it is done, you will not experience the slow initial opening of your files.

To access the Conversion Wizard, click the Microsoft **Start** button, select **Programs**, and select the **SolidWorks 99** program group. Click **Conversion Wizard**.

When the conversion utility begins, it offers you the choice of backing up all of your files before the conversion. If you choose to backup your SolidWorks files, the Conversion Wizard copies the files to a folder named “Solidworks Conversion Backup.” The wizard asks you for the location of the files to be converted, and leads you through the simple process.

At the end of the conversion process, two report files exist in the folder to which you directed the conversion. **Conversion Wizard Done.txt** contains a list of files that converted; **Conversion Wizard Failed.txt** contains a list of files that did not convert. Double-click the file names to open them in Notepad.

## Visiting the SolidWorks Web Site

---

If your PC is connected to the Internet, you can visit the SolidWorks Web site after your installation is complete. You can learn more about the SolidWorks company and product by clicking in the **Help** menu on the main SolidWorks window.

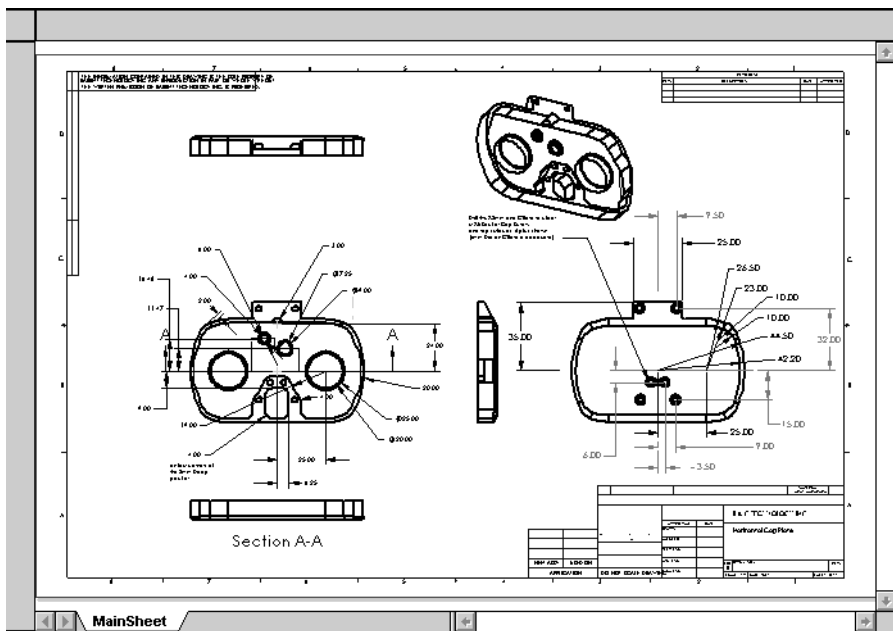
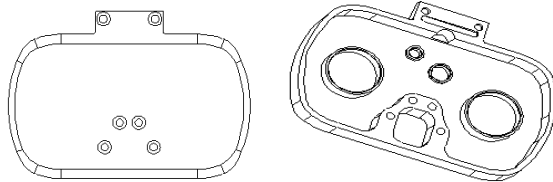
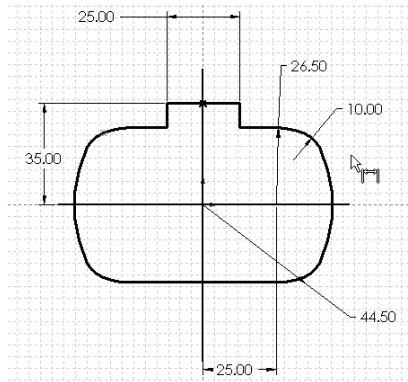
### *To access the SolidWorks Web site:*

- 1 Click **Help, About SolidWorks 99**.
- 2 Click the **Connect** button to visit the SolidWorks Web site.  
You can learn about **What's New, Technical Support, VARs and Distributors**, and you can visit the SolidWorks **Design Gallery**, among other things.

## Basic Concepts

□ A SolidWorks model consists of parts, assemblies, and drawings.

- Typically, you begin with a sketch, create a base feature, and then add more features to your model. (You can also begin with an imported surface or solid geometry.)
- You are free to refine your design by adding, changing, or reordering features.
- Associativity between parts, assemblies, and drawings assures that changes made to one view are automatically made to all other views.
- You can generate drawings or assemblies at any time in the design process.



❑ **The SolidWorks 99 application lets you customize functionality to suit your needs.**

Click **Tools, Options** on the main menu bar to display tabbed pages of available options. The options pages are described in detail in Appendix A; the following describes just a few ways in which you can customize the SolidWorks 99 application:

<b>Color</b>	Colors for lines, faces, features, drawing, and system items
<b>Detailing</b>	The dimensioning standard, precision, and tolerances; styles for arrows, witness lines, balloons, or virtual sharp representation; font size and style
<b>Drawings</b>	Drawing sheet size, scale, projection type, default edge display, automatic update of BOM and model views, and inferencing behavior
<b>Edges</b>	Display of edge types, repainting and highlighting behavior, edge selection options
<b>External References</b>	Preferences for the way part, assembly, and drawing documents are opened and referenced; definition of search paths for external documents, palette features, and palette parts
<b>General</b>	Optional behaviors when in a sketch or model, FeatureManager design tree behavior, angle increment and speed of view rotation, backup and AutoRecover options
<b>Grid/Units</b>	Grid display, spacing, and snap behavior; length unit types and angular unit types; spin box increment values
<b>Line Font</b>	Style and weight of lines for selected edge types
<b>Performance</b>	Choices about model display quality, curvature generation, and behavior of lightweight parts in assemblies

- ❑ **The Registry Copy Tool lets you share customized options with other SolidWorks users.**

After you customize your SolidWorks 99 software, you can use the **Copy Options Wizard** to copy the options information set by one user and specify the same options on the machines of other users.

**To use the Copy Options Wizard:**

- 1 From the **Programs** list, select **SolidWorks 99, Copy Options Wizard**.
- 2 To capture your SolidWorks options information from the registry, click **An existing user** and select your name from the list. Click **Next**.
- 3 Select one of the following options:
  - **Users or Computers on the network.** As a system administrator on Windows NT, you can copy the SolidWorks options directly to other selected users or computers on the network.
  - or –
  - **Create a Registry file.** The SolidWorks registry options of the current system are copied to a **swOptions.reg** file that you can use with standard Windows NT, Windows 95, or Windows 98. Just copy this **swOptions.reg** file to another computer and double-click the file. The registry options are automatically copied to the registry of the new computer.


- ❑ **The SolidWorks application saves your work for you.**

The **AutoRecover** option automatically saves information about your active part, assembly, or drawing document so you will not lose your work in case of a system crash. To set this option, click **Tools, Options, General** and select **Save AutoRecover info every (n) changes**. Specify the number of changes that should occur before information is automatically saved. See Appendix A, "General Options" on page A-13.

## Getting Help

---

If you have questions while you are using SolidWorks 99, you can find answers in several ways:

- ❑ SolidWorks online help provides assistance while you are working.
  - To get help about a specific, active dialog box, click the **Help** button in the dialog box or press the **F1** key.
  - Click  or **Help, SolidWorks 99 Help Topics** on the main toolbar to access the online help system.
- ❑ For helpful hints, click **Help, Tip of the Day**. To see a tip each time you start SolidWorks 99, click **Show Tips at Startup** in the **Tip of the Day** dialog box.



- ❑ **Tooltips** are available to provide information about tools on toolbars or dialog boxes. When you hold the pointer over a tool for a brief time, a tooltip appears to display the name of the tool.
- ❑ As you move the pointer over an tool or click a menu item, the **Status Bar** at the bottom of the SolidWorks window provides a brief description of the function of the tool or menu item.
- ❑ The *SolidWorks 99 Tutorial* provides much detailed information about using SolidWorks to create parts, assemblies, and drawings.


## Opening a SolidWorks Part, Drawing, or Assembly

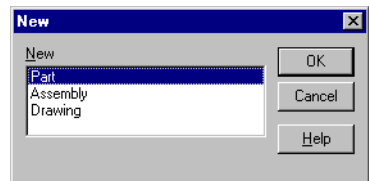
---

This section describes how to open a new or existing part, drawing, or assembly document.

### Opening a New SolidWorks Document

*To open a new part, drawing, or assembly document:*

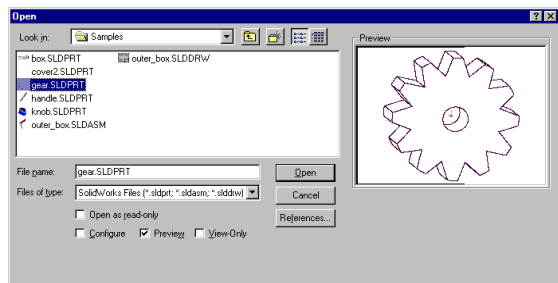
- 1 In a SolidWorks window, click  on the Standard toolbar, or click **File, New**.
- 2 From the **New** menu, select **Part, Assembly, or Drawing** and click **OK**.



### Opening an Existing SolidWorks Document

*To preview and open an existing part, drawing, or assembly document:*

- 1 In a SolidWorks window, click **File, Open**. Use the browser to select the part, drawing, or assembly.
- 2 Select **Open as read-only** if you want the document to be opened in read-only mode. This allows other users to have write access to the part at the same time.
- 3 Select the **Preview** check box to see a preview of the part.
- 4 Select **Configure** to open the part in a specified configuration. For more information about configurations, see Chapter 8, “Working with Configurations.”



- 5 Click **View-Only** to open the document for viewing. (Only documents saved in SolidWorks 98, SolidWorks 98Plus, or SolidWorks 99 may be opened in **View-Only** mode.) If you are in a part or assembly document, you can change to editing mode by right-clicking in the graphics area and selecting **Edit**.
- 6 If you are opening an assembly or drawing document, you can change the path from which referenced parts are taken by clicking **References**. In the dialog box that appears, select and type the **New pathname**, and click **Replace**. See **Editing Referenced File Locations** on page 6-14.
- 7 Click **Open** to open the document.

### Multi-Threaded Retrieval

Documents saved in SolidWorks 98, SolidWorks 98Plus, and SolidWorks 99 are retrieved in multi-threaded mode.

When you retrieve a large part, drawing, or assembly document, the document immediately displays in a view-only state while the actual document and all its components are being retrieved in the background. During the view-only state, you can use all functions supported in the SolidWorks Viewer (Zooming, Rotation, and so on.), but you cannot switch to another document or start opening another document. After the retrieval is completed, SolidWorks changes to the normal edit state.

### Opening an Existing Part from Explorer

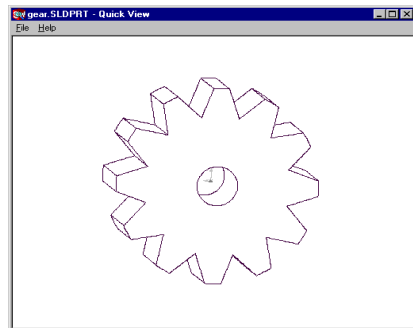
You can preview and open a part, drawing, or assembly document directly from the Microsoft Explorer.

#### *To view the part, drawing, or assembly without opening the document:*

Right-click the name of the part, drawing, or assembly in Explorer, and select **Quick View**.

**Quick View** displays the part in a **SolidWorks Viewer** window.

You can also view thumbnail images of SolidWorks parts and assemblies in the Explorer as well as in the **File, Open** and **File, Save As** dialog boxes. The graphic is based on the view orientation of the model when the document was saved. To enable this functionality, click **Tools, Options**, and select the **General** tab. Select the option **Show thumbnail graphics in Windows Explorer**. See Appendix A, "General Options" on page A-13.



**To open a part, a drawing, or an assembly from Explorer, do one of the following:**

- Double-click the name of the part, drawing, or assembly document in Explorer.
- Right-click the name of the part, drawing, or assembly document in Explorer and select **Open** from the shortcut menu.
- Drag and drop any SolidWorks document from Explorer into an empty area of the SolidWorks window, not occupied by another document window.

### **Dragging and Dropping Files**

- Drag and drop a part from Explorer to an open SolidWorks assembly window to add an instance of the part or sub-assembly to the assembly.
- Drag and drop a part or assembly from Explorer to an open and empty SolidWorks drawing document to create a Standard 3 View drawing of the part or assembly.
- You can drag and drop hyperlinked SolidWorks part files from Internet Explorer, version 4.0 or later, to:
  - the Feature Palette window
  - a new, empty part
  - a drawing or assembly document
  - an empty area of a SolidWorks window

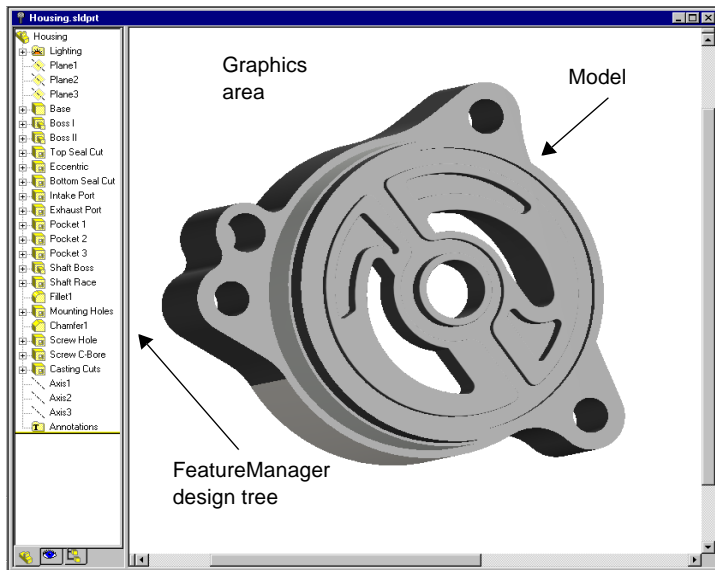
If you drop the file into a drawing or an assembly, you are prompted to name the new file on your local disk.

## SolidWorks Terms

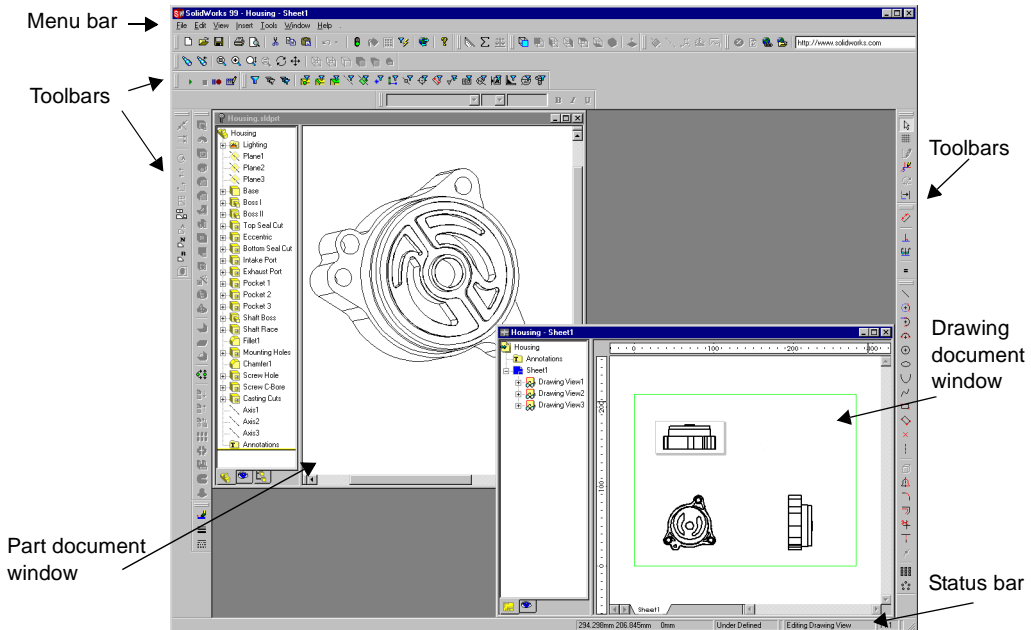
SolidWorks document windows have two panels:

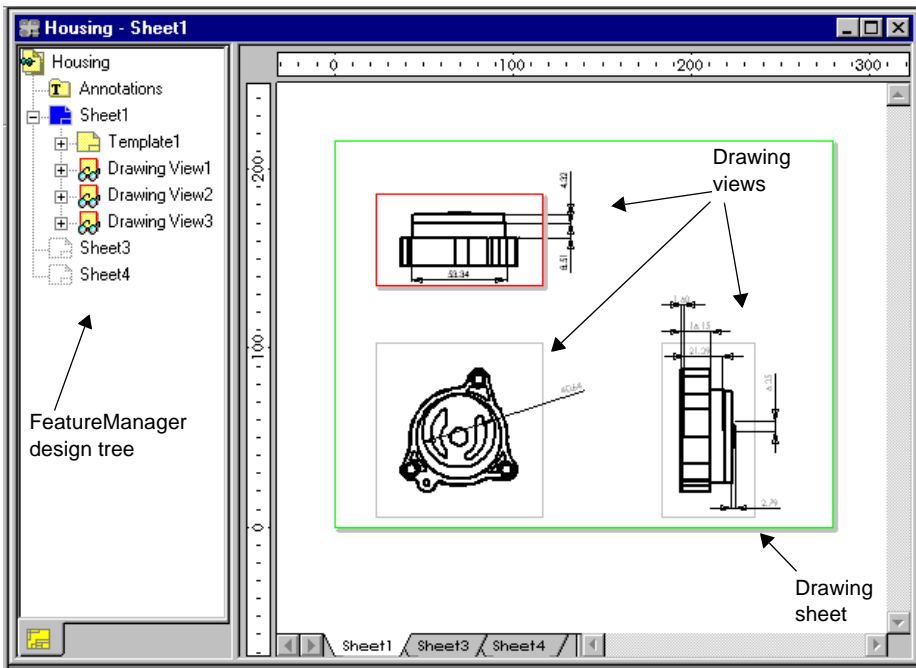
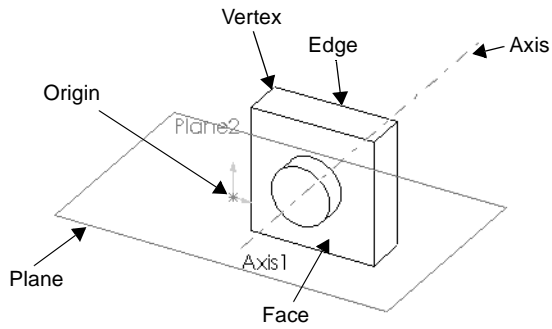
A *FeatureManager design tree*, which lists the structure of the part, assembly, or drawing. For more information about the FeatureManager design tree, see page 1-18.

A *graphics area*, where you create and manipulate the part, assembly, or drawing.



This section illustrates some common SolidWorks terms.





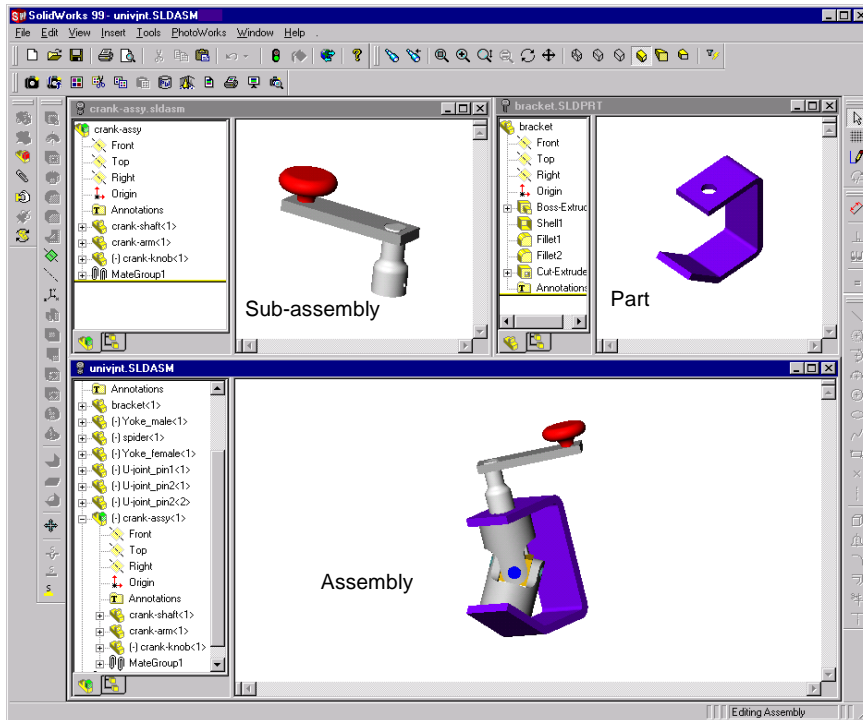
Sheet tabs


## Document Windows

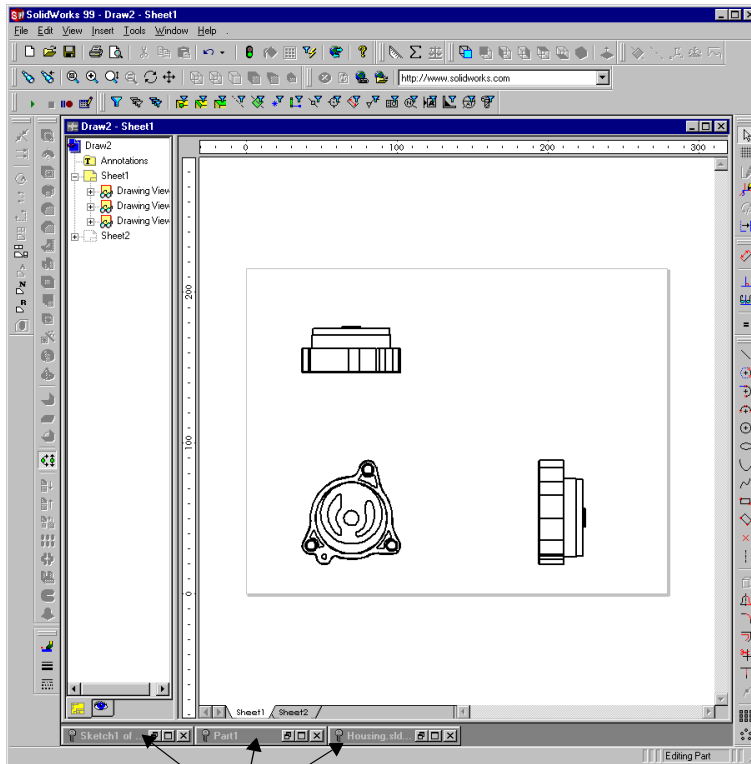
In the SolidWorks application, each part, assembly, and drawing is referred to as a document, and each document is displayed in a separate window. (Each drawing document can contain multiple drawing *sheets*, though.)

You can have multiple part, assembly, and drawing document windows open at the same time. Also, you can have multiple views of the same document visible at the same time.

To arrange the windows, you can drag them by the title bar, and resize them by dragging a corner or border. Also, from the **Window** menu, you can select **Cascade**, **Tile Vertically**, or **Tile Horizontally**.



To organize your SolidWorks window, you can iconize open documents. Click the **Iconize**  symbol in the upper right corner of the document border. An icon appears in the lower part of the SolidWorks window. If the icon is not visible, it may be behind another open document. Resize any open documents as necessary. Click **Window, Arrange Icons** to arrange them at the bottom of the SolidWorks window.



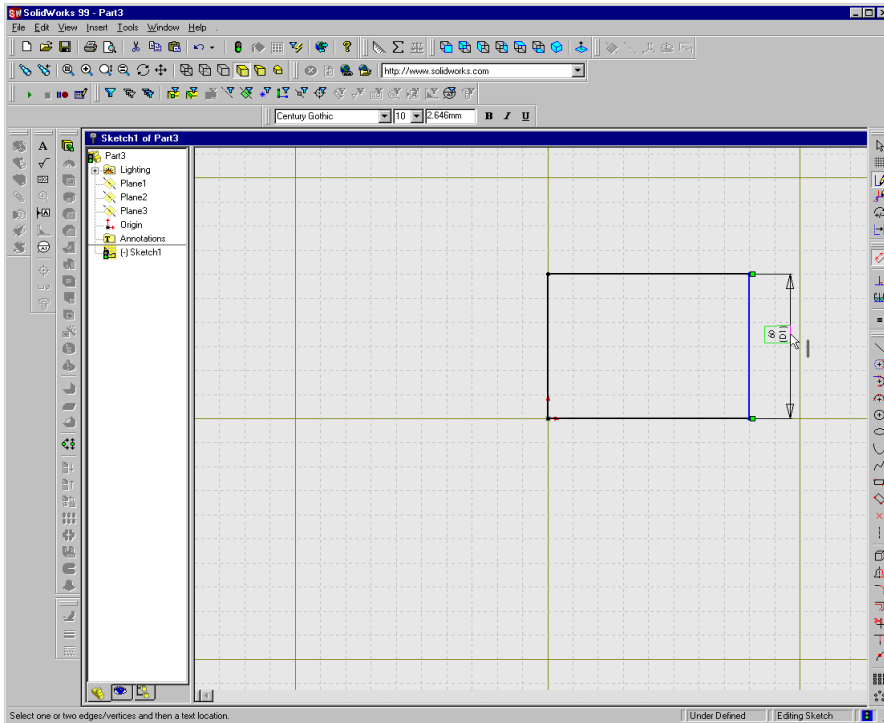
Document icons

Click **Window, Close All** to close all open documents. You are prompted to **Save** unsaved documents.

## Toolbars

The toolbar buttons are shortcuts for frequently used commands. Most of the available toolbars are displayed in this illustration of the SolidWorks initial screen, but your SolidWorks window probably will not be arranged this way. You can customize your toolbar display in a way that is convenient for you.

Some toolbars are always displayed; other toolbars display automatically when you open a document of the related type. For example, when you open an assembly document, the assembly toolbar displays.



### **To display or hide individual toolbars:**

Click **View, Toolbars**, or right-click the SolidWorks window frame.

A list of all the toolbars is displayed. The toolbars with a check mark beside them are visible; the toolbars without a check mark are hidden. Click the toolbar name to turn its display on or off.

To make changes to the toolbars, click **Tools, Customize**. You can move toolbar buttons to different toolbars, change menus, or reset shortcut keys. See **Customize and Arrange Toolbars** on page 1-17.



## Customize and Arrange Toolbars

### To customize toolbars:

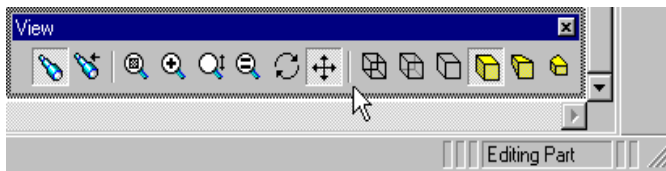
Click **Tools, Customize**, or right-click the SolidWorks window frame and click **Customize** at the bottom of the toolbar list.

- On the **Toolbars** tab, you can do the following:
  - select individual toolbars to display or hide
  - display the toolbar buttons in a larger size
  - display or hide the tooltips that appear when the pointer moves over a button
- On the **Commands** tab, you can do the following:
  - rearrange the command buttons on toolbars
  - move buttons from one toolbar to another
  - place copies of buttons on more than one toolbar
  - delete buttons that you never use by dragging them from the toolbar into the graphics area
- On the **Menus** tab, you can do the following:
  - customize the SolidWorks menus to suit your style of working
  - rename, relocate, or remove commands from the menus
- On the **Keyboard** tab, you can do the following:
  - customize the keyboard shortcut keys to suit your style of working
  - assign new shortcut keys, remove shortcut keys, or reset the shortcut keys to their original setting
- On the **Macros** tab, you can assign macros to buttons that you can place on any toolbar. For information about creating macros and assigning them to toolbar buttons, refer to Appendix B, "Recording and Playing SolidWorks Macros" on page B-2.

### To arrange toolbars:

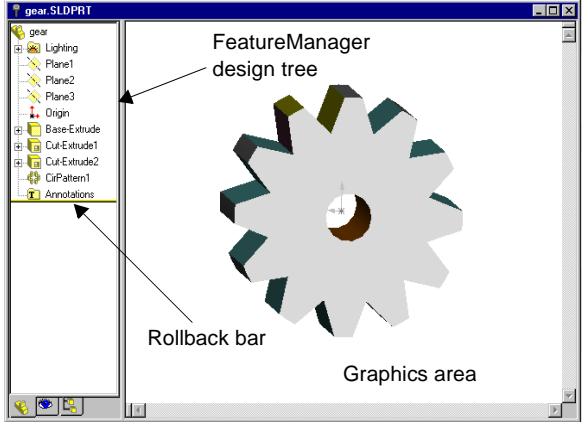


Click on the space *between* the buttons and drag. If you drag a toolbar to an edge of the SolidWorks window, the toolbar docks to the edge automatically.

If you drag a toolbar away from the window edge, it becomes a floating palette.






## The FeatureManager Design Tree

The FeatureManager design tree makes it easy to:

- Select items in the model by name.
  - Adjust the kind and amount of lighting that illuminates a shaded part or assembly. See the section on **Lighting** beginning on page 5-27.
  - Select a plane to sketch on.
  - Suppress or hide selected features.
- 
- The screenshot shows the SolidWorks interface for a gear part. On the left is the FeatureManager design tree, which is a hierarchical list of features. The features listed are: gear, Lighting, Plane1, Plane2, Plane3, Origin, Base-Extrude, Cut-Extrude1, Cut-Extrude2, CirPattern1, and Annotations. The 'Annotations' folder is currently selected. On the right is the Graphics area, which displays a 3D model of a gear. The gear is a white, multi-toothed part with a central hole. A 'Rollback bar' is visible at the bottom of the Graphics area, and the 'Graphics area' label is at the bottom right. The title bar of the window reads 'gear.SLDPRT'.
- Temporarily roll the model or assembly back to an earlier state using the *rollback* bar. See **Rollback** on page 5-15.
  - Do a text search of the FeatureManager design tree for an item. Right-click the name of the part or assembly at the top of the tree and select **Go To** from the shortcut menu. Type the name of the item (or a part of the name) in the box and click the **Find Next** button. The name scrolls into view.
  - Find an item in the FeatureManager design tree by right-clicking it in the graphics area and selecting **Go To Feature (in Tree)**.
  - Identify and change the order in which features are rebuilt.
  - Display the dimensions of a feature by double-clicking the feature's name.
  - Rename features by slowly double-clicking the name and then typing a new name (a standard Windows behavior). You can also right-click a name, select **Properties**, and type a new name in the **Name** box.
- 
- NOTE:** Feature names are not case sensitive: “Boss1” and “boss1” are the same. Feature names may not contain the @ character.
- 
- Add a new equation, edit, or delete an equation by right-clicking the **Equations** folder , and selecting the action you want. See **Equations** on page 5-9.
  - Switch between the features list and the list of named configurations using the tabs at the bottom of the FeatureManager design tree. The tabs also have tool tips.
  - Control the display of dimensions and annotations by right-clicking the Annotations folder  and selecting the options. See **Displaying Annotations in Parts** on page 5-22.

## Symbols and Conventions

- The FeatureManager design tree gives you information about any parts or features with *external references*. (An external reference is a dependency on geometry that exists in another document.) For more about external references, see page 5-18 and page 7-2.
  - If a part contains any features with external references, the part name at the top of the tree list is followed by →. The name of any feature with external references is also followed by →.
  - If an external reference is currently out of context, the feature name and the part name are followed by ->?
- The FeatureManager design tree uses the following conventions:
  - A ⊕ symbol to the left of an item's icon indicates that it contains associated items, such as sketches. Click the ⊕ to expand the item and display its contents.
  - Sketches in the FeatureManager design tree are preceded by (+) if they are over defined; they are preceded by (-) if they are under defined; they are preceded by (?) if the sketch could not be solved. For more information about over defined and under defined sketches, see **Fully Defining Sketches** on page 2-44. (There is *no* prefix if the sketch is fully defined.)
  - Assembly components in the FeatureManager design tree are preceded by (+) if their position is over defined; they are preceded by (-) if their position is under defined; they are preceded by (?) if their position could not be solved; they are preceded by (f) if their position is *fixed* (locked in place). If the position is fully defined, there is no prefix.
  - Assembly mates are preceded by (+) if they are involved in over defining the position of components in the assembly; they are preceded by (?) if they could not be solved.
  - In an assembly, each component is followed by a number in angle brackets <n> that increments with each instance of that component.
- The tabs at the bottom of the FeatureManager design tree indicate the current FeatureManager function:
  -  A part or a sketch document is open for editing, viewing. See Chapters 2, 4, and 5, about “Sketching,” “Creating Features,” and “Working with Parts.”
  -  An assembly is open for editing, adding components, creating configurations, and viewing. See Chapter 6, “Working with Assemblies.”
  -  The PropertyManager functionality is in use. For more information, See page 2-45 in Chapter 2, “Sketching.”



The Configuration tab lets you create, select, and view the configurations of a part or assembly. See Chapter 8, “Working with Configurations.”



A drawing document is open for viewing or editing. See Chapter 9, “Drawings.”

## FeatureManager Design Tree Options

Select options for the FeatureManager design tree by clicking **Tools, Options** and selecting the **General** tab. (For more information, see "General Options" on page A-13.) Select from the following options:

- **Scroll selected item into view.** Scrolls to display the text that is related to the selected items in the graphics area.
- **Name feature on creation.** When you create a new feature, the feature’s name is selected and ready for you to type a new name.
- **Arrow key navigation.** Lets you use the arrow keys to traverse the FeatureManager design tree, and expand or collapse the design tree and its contents.
- **Dynamic highlight.** The geometry in the graphics area (edges, faces, planes, axes, and so on) highlights when the pointer passes over the item in the FeatureManager design tree.

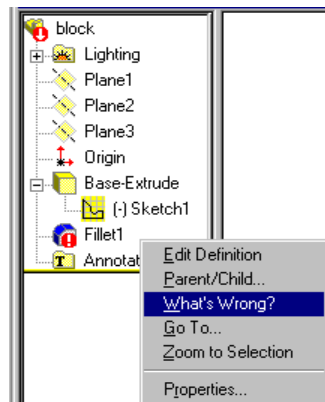
## What’s Wrong?

The SolidWorks application offers a “What’s Wrong” functionality, that allows you to view any errors that occur when rebuilding a part or assembly. A red circle with a down-pointing arrow next to the part or assembly name at the top of the FeatureManager design tree alerts you that there is a problem. An exclamation mark (!) indicates the item responsible for the error.

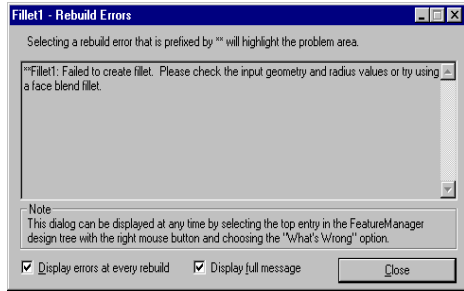
Right-click the sketch, feature, part, or assembly name and select **What’s Wrong** to display the error.

Some common errors in rebuilding include:

- Dangling dimensions or relations – dimensions or relations to an entity that no longer exists
- Features that cannot be rebuilt, such as a fillet that is too large




The first time an error occurs, the **Rebuild Errors** dialog box is displayed. To turn off the automatic display of errors, clear the **Display errors at every rebuild** check box in the **Rebuild Errors** dialog box. You can also display the dialog box by right-clicking on the name of the part or assembly in the FeatureManager design tree and selecting **What's Wrong**.

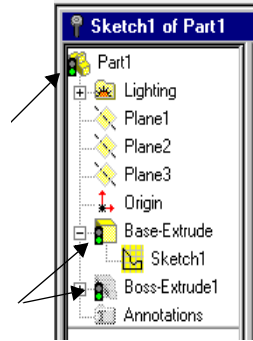


**NOTE:** The **Display errors at every rebuild** check box only affects the current session.

## Rebuild Symbol

If you make changes to a sketch or part that require the rebuild of the part, a rebuild icon  is displayed beside the part's name as well as beside the feature that requires the rebuild. Look for the rebuild icon on the Status Bar, also.



The rebuild symbol also displays when you edit a sketch. When you exit the sketch, the part rebuilds automatically.



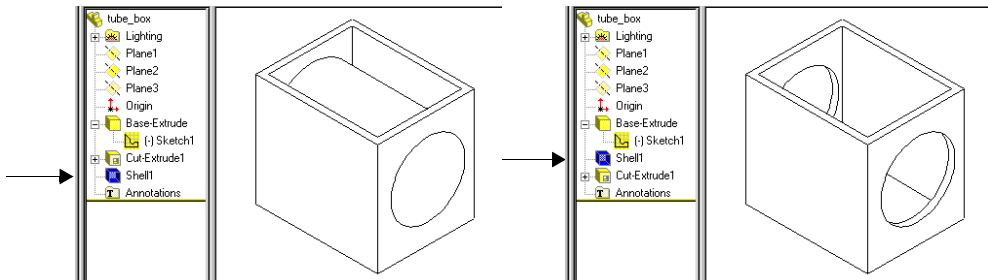
## Drag and Drop Features

The SolidWorks software supports several drag and drop operations for features.

- ❑ **Reordering features.** You can change the order in which features are rebuilt by dragging them in the FeatureManager design tree. Place the pointer on a feature name, press the left mouse button, and drag the feature name to a new position in the list. (As you drag up or down the tree, each item that you drag over highlights. The feature name that you are moving drops immediately below the currently highlighted item when you release the mouse button.)

If the reorder operation is legal, a  pointer appears; if it is not legal, a  pointer appears.

In the following example, the **Cut-Extrude1** was cut before the **Shell1** feature was added. Later, **Shell1** was dragged and dropped before **Cut-Extrude1**.



- ❑ **Moving and copying features.** You can also move features by dragging them in the model. For example, you can move a hole to a different face.  
You can copy or move a fixed-radius fillet or a chamfer using drag and drop.  
To create a copy of the feature, hold down the **Ctrl** key while you drag.  
For more information about moving and copying features, see **Moving and Resizing Features** on page 5-5 and **Copying Features** on page 5-7.
- ❑ **Dragging and dropping between open documents.** You can drag a part name from the FeatureManager tree of an open part file to insert it in an open assembly document. And, you can drag a part or assembly name from the FeatureManager tree to a drawing document.

## Shortcut Menus

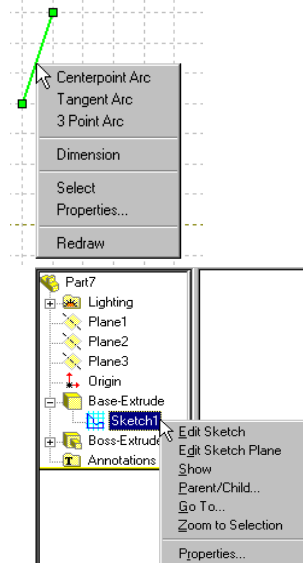
---

Whether you are working with a sketch, a model, an assembly, or a drawing, you have access to a wide variety of tools and commands from the shortcut menu accessed by pressing the right-mouse button. This gives you an efficient way to do your work without continually moving the pointer to the main menus or the toolbar buttons.

As you move the pointer over geometry in the model or over items in the FeatureManager design tree, right-clicking pops up a shortcut menu of commands that are appropriate for whatever you clicked on.

For example, with the shortcut menu, you can:

- Select a sketch tool
- Open and close sketches
- Change or view the properties of an item
- Give a new name to a feature or dimension using the **Properties** dialog
- **Hide** or **Show** a sketch, plane, axis, or assembly component
- Open an assembly component for editing
- Access the dimension tools and an annotations menu when in a drawing
- Find an item in the FeatureManager design tree by right-clicking it in the graphics area and selecting **Go To Feature (in Tree)**.
- Change view mode (See **The View Toolbar** on page 1-26.)




## Selection Methods

---

### Selecting from the Graphics Area

Most commands require you to make selections. For example, to create a fillet, you have to select the model edges or faces you want to fillet.

- ❑ **Selecting.** Click the  button (the selection tool), then click the item you want to select.


Notice that items change color as the pointer passes over them. This *dynamic highlighting* helps you locate the item to select.

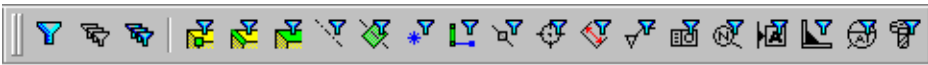
---

**NOTE:** For information about turning dynamic highlighting *off* or *on*, see "Edges Options" on page A-10.




---

- ❑ **Selecting with the Selection Filter.** To make it easier to select specific items, you can set the **Selection Filter** to the kind of item that you want to select: faces, edges, and vertices in parts. You can also set it for reference geometry, sketch entities, or dimensions and annotations. With the filter set, the kinds of items that you specified are identified when you pass the pointer over them. Sometimes they are highlighted, and sometimes the pointer changes shape. This makes it easy for you to select only the items that you intend to select.

To toggle the display of the Selection Filter toolbar, click  on the Standard toolbar, or press **F5**.



The first three buttons allow you to specify the behavior of the Selection Filter toolbar.


-  Turn all the selected filters on or off. You can also press **F6**.
-  Clear all selected filters.
-  Select all of the filters.

The rest of the buttons are filters. Select the filters that match the items you want to select in the graphics area.

#### Selection Filter hot keys:

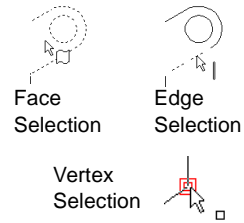
- Filter for edges      Press **E**
- Filter for faces      Press **X**
- Filter for vertices      Press **V**

---

**NOTE:** As a reminder to you, while a **Selection Filter** is active, the selection pointer has a small filter icon attached to it. 

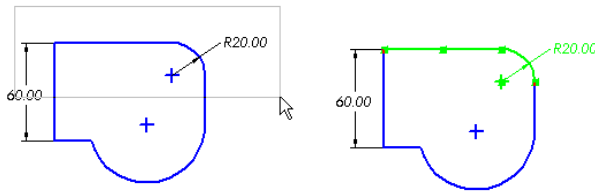
---

- ❑ **Face and Edge Selection pointers.** To help you select items, the pointer symbol changes to indicate the kind of item that the pointer is currently passing over. When you pass the pointer over a face, the pointer symbol is shaped like a flag; when it is over an edge, the symbol is a vertical bar. When you pass the pointer near a vertex, the symbol is a square and the vertex displays two concentric squares.

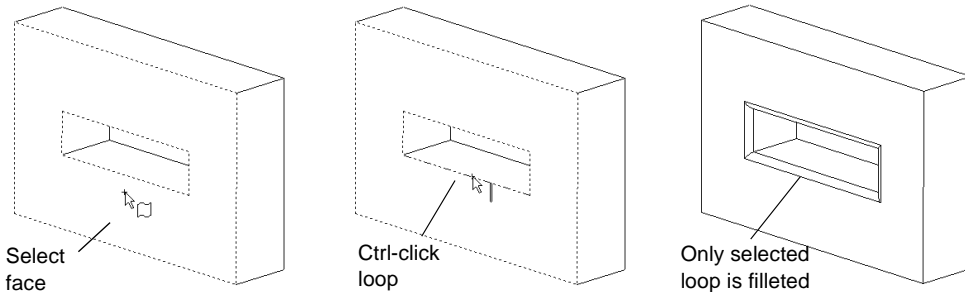


- ❑ **Selecting the midpoint of an entity.** For easy selection of the midpoint of an edge, right-click the edge and click **Select Midpoint**. A selection point appears in the middle of the entity.
- ❑ **Selecting multiple items.** To select more than one item, hold down the **Ctrl** key while you click the items. However, if you are selecting multiple items while in a dialog box, it is usually not necessary to hold the **Ctrl** key.

- ❑ **Selecting by dragging.** In a sketch or drawing, you can drag a selection rectangle around items to select them. Press the left mouse button and drag the pointer across the area forming a selection rectangle. Entities that are completely within the rectangle are selected and highlighted.



- ❑ **Selecting loops.** If you select a face, all *loops* (closed collections of connecting edges) on the face are selected, too. To select individual loops, click the face, then hold down the **Ctrl** key and click the loops you want to select. (This deselects the face and leaves the loops selected.)




- ❑ **Changing a selection.** Some dialog boxes show you a list of the selected items. You can change the selection while the dialog box is open by clicking a selected item a second time to clear it. To start over, right-click in the graphics area and choose **Clear Selections**.



## Selecting Hidden or Coincident Items

Sometimes the item you want to select is *behind or coincident with* another item.

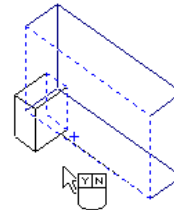
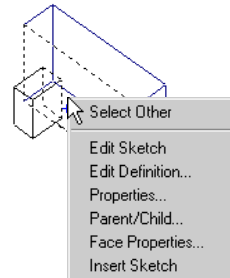
### To select a hidden item:

- 1 Right-click where you want to make a selection.
- 2 Click **Select Other** on the shortcut menu.  
Notice the yes/no pointer .  
Clicking the *right* button (**N**) highlights each of the items under the point where you clicked, in turn.
- 3 When the item you want is highlighted, click the *left* button (**Y**) to select that item.

---

**NOTE:** You can make selections by clicking items in the graphics area or the FeatureManager design tree, whichever is more convenient.

---



## Selecting from the FeatureManager Design Tree

There are several ways to select directly from the FeatureManager design tree.







- You can select multiple consecutive items in the FeatureManager design tree by holding the **Shift** key while you select. Click the first item, hold down **Shift**, and click the last item.
- You can select multiple items in the graphics area, or select non-consecutive items in the FeatureManager design tree by holding down the **Ctrl** key while you select.
- You can select features, sketches, planes, and axes in the model by clicking on their names in the FeatureManager design tree.








## Viewing and Manipulating Parts, Assemblies, and Drawings



### The View Toolbar

The View toolbar provides tools for manipulating SolidWorks parts, drawings, and assemblies. (Not all of these tools are appropriate for drawings, however.)



Tool	Description
 <b>View Orientation</b>	Displays the <b>Orientation</b> dialog box. (You can also display the <b>Orientation</b> dialog box by pressing the <b>Space Bar</b> .)
 <b>Undo Last Change</b>	Returns to the previous view. You can undo the last 10 view changes.
 <b>Zoom to Fit</b>	Rescales the view so the entire part, drawing, or assembly is visible.
 <b>Zoom to Area</b>	<p>Zooms in on a portion of the view that you select by creating a bounding box.</p> <ol style="list-style-type: none"> <li>1. Place the pointer where you want to place the first corner of the box.</li> <li>2. Drag the pointer diagonally to the opposite corner of the bounding box.</li> <li>3. Release the pointer. The image enlarges proportionally to the size of the bounding box.</li> <li>4. To resize the image back to fit the screen, click <b>Zoom to Fit</b>.</li> </ol>
 <b>Zoom In/Out</b>	<p>Dynamically changes the scale of the image.</p> <ul style="list-style-type: none"> <li>• To enlarge the image, drag upward.</li> <li>• To reduce the image, drag downward.</li> </ul>
 <b>Zoom to Selection</b>	<p>Zooms in on a portion of the model, assembly, or drawing that you select.</p> <p>Select an entity to which you want to zoom, or hold the <b>Ctrl</b> key and select several entities, then click <b>Zoom to Selection</b>.</p> <p>To zoom back out, click <b>Zoom to Fit</b>.</p>

 <b>Rotate View</b>	<p>Dynamically turns the part or assembly image around a view center as you move the pointer. (Not for drawings.)</p> <ul style="list-style-type: none"> <li>• To rotate the image on a vertical axis, move the pointer left to right.</li> <li>• To rotate the image on a horizontal axis, move the pointer up and down.</li> <li>• To rotate the image diagonally, move the pointer diagonally.</li> <li>• To rotate the image about a specific point, line, or plane, select the <b>Rotate</b> tool, select the entity to rotate about, and use the pointer to rotate, as usual. (Click an empty area to deselect the point, line, and so on.)</li> </ul> <p>You can also use the keyboard arrow keys to rotate the image.</p>
<p><b>NOTE:</b> To change the speed of the view rotation, click <b>Tools, Options</b>, and select the <b>General</b> tab. Use the slider to change the speed of rotation with the pointer; you can change the angle increment used when rotating with the arrow keys.</p>	
 <b>Pan</b>	<p>Dynamically moves the image. Drag the pointer around on the screen.</p>
 <b>Wireframe</b>	<p>Displays all the edges of the part or assembly.</p>
 <b>Hidden in Gray</b>	<p>Displays edges that are obscured from view in a light gray line. Visible edges are displayed normally.</p>
 <b>Hidden Lines Removed</b>	<p>Displays only those lines that are visible at the angle the model is rotated. Obscured lines are removed.</p>
<p><b>NOTE:</b> To change the default display color of lines in the above three view types, click <b>Tools, Options, Color</b>. Select the line type, click <b>Edit</b>, and select a color from the color palette. Click <b>OK</b>.</p>	
 <b>Shaded</b>	<p>Displays a shaded image of the model. This is the default view mode.</p>
<p><b>NOTE:</b> To change the default color of the shaded part, click <b>Tools, Options, Color</b>. Select <b>Shading</b>, click <b>Edit</b>, and select a color from the color palette. Click <b>OK</b>.</p>	
 <b>Perspective</b>	<p>Displays a perspective view of the model. A <i>perspective</i> view is the view normally seen by the eye or a camera. Parallel lines recede into the distance to a vanishing point. Perspective may be used in combination with any of the view modes.</p>

 <p><b>Section View</b></p>	<p>Creates a section view of the model or assembly for display purposes. You cut through the model or assembly using one of the following:</p> <ul style="list-style-type: none"> <li>• one or more standard planes (Plane1, 2, or 3)</li> <li>• one or more planes that you create for the purpose (Click <b>Insert, Reference Geometry, Plane</b>, and create a new plane.) See <b>Creating a Construction Plane</b> on page 3-2.</li> <li>• one or more planar faces on the model</li> <li>• the viewing plane (The invisible plane that is parallel to the screen you are looking through. The initial position for this plane is the center of the part body bounding box.) See <b>Section View of a Model</b> on page 5-23.</li> </ul>
 <p><b>Toggle Selection Filter Toolbar</b></p>	<p>Shows or hides the Selection Filter toolbar. (You can also show or hide the Selection Filter by pressing the <b>F5</b> key.)</p>

---

**NOTE:** For information about other SolidWorks toolbars, click **Help, SolidWorks 99 Help Topics**. Select **SolidWorks Fundamentals**, and click **Toolbars**.

---

## The Standard Views Toolbar

The Standard View toolbar provides tools for quickly moving your sketch, model or assembly to any of the standard views. To change the view, click a different view tool.



Front View



Back View



Left View



Right View



Top View



Bottom View



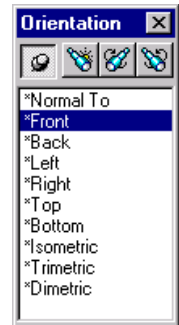
Isometric View




Normal To – Select a plane or planar face

## Using Named Views

You can display a part or assembly using named views with the **Orientation** box.



### To display the Orientation box:

- Click  on the View toolbar.
- Click **View, Orientation**.
- Press the **Space Bar**. The **Orientation** box appears under the pointer.

To switch quickly between named views, double-click the name of the view that you want to see.

Use the buttons on the **Orientation** box as follows:




The **Pushpin** keeps the **Orientation** box open and on top of all other windows. Drag the dialog box to a convenient location on your screen and click the pushpin icon to hold it in place.

If the pushpin is holding the **Orientation** box open when you exit the SolidWorks application, it is restored to the same location when you run the application again.



The **New View** tool lets you make additions to the standard list of view orientations. You can rotate the model or change the zoom, then click the **New View** tool. Type a descriptive name for the view, and click **OK**. The name appears at the top of the list of views, and you can rotate your model or assembly to the view by double-clicking the view name.



The **Update Standard Views** tool lets you change the orientation of all the standard views. Double-click on one of the named views to select the new orientation, then click the name of the standard view you want to assign to the current orientation of the model. For example, if you want what is currently the **Left** view to become the **Front** view, double-click **Left**, single-click **Front**, and then click . This updates all of the standard views so they are relative to this view.



Click the **Reset Standard Views** tool to return all standard views to their default settings.

### To return your model to a previous view:

Click **Previous View**  on the View toolbar.

–or–

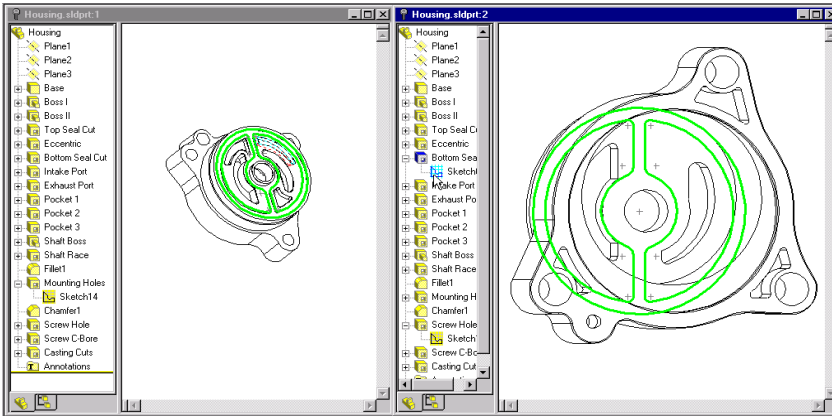
Click **View, Modify, Undo View Change**, or press **Ctrl+Shift+z**.

You can undo the last 10 view changes.

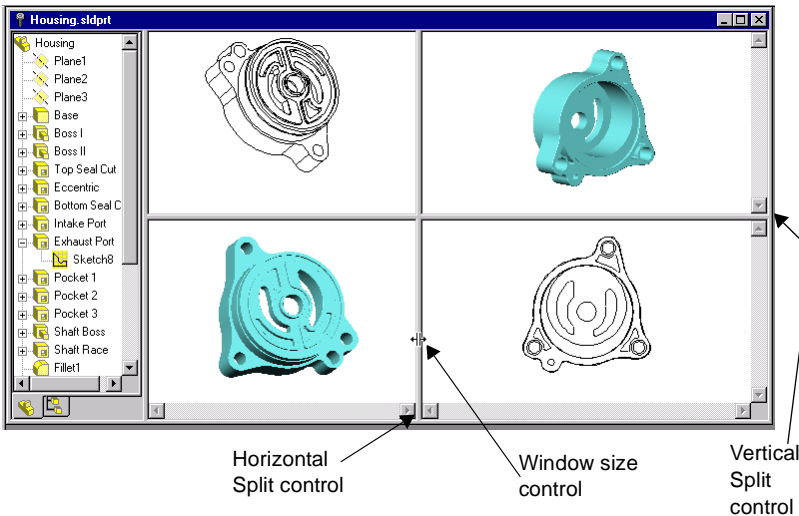
## Multiple Views

You can open additional views of the same document. Selecting an item in one view selects it in all views. For example, when creating a fillet you could select edges on the front of the model in one view and edges on the back in another view.

- ❑ Click **Window, New Window** to open another view in a new window.



- ❑ Drag the horizontal or vertical **Split** controls to split the window into two or four panes. You can zoom, rotate and set the view mode for each of these views independently.



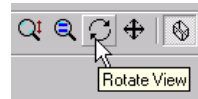
Click in the view, then set the view options.

## Time Saving Tips

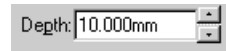
---

The following tips may help you save time.

- ❑ **Save snapshots.** Use the **Save As** command to save successive “snapshots” of your design under different names.
- ❑ **Use tooltips.** To identify a button on a toolbar, just hold the pointer over the button; a *tooltip* will soon appear.



- ❑ **Use spin box arrows to change values.** If a value box has arrows next to it, you can increase or decrease the value by clicking on the arrows. (You can change the **Spin Box Increments** value with the **Tools, Options** command. Click the **Grid/Units** tab.)



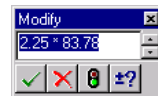
- ❑ **Modify dimensions with the spin box.** You can change the dimension values by clicking on the arrows. In addition, you can use the row of buttons below the value window to do any of the following:



- Save the current value and exit.
- Restore the original value and exit.
- Rebuild the model with the current value.
- Reset the **Spin Box Increments** value.

If the model has multiple configurations, you can apply the new value to **This configuration** only, or to **All configurations**.

- ❑ **Use the Modify box as a calculator.** Type values and arithmetic symbols directly into the box to calculate the dimension.
- ❑ **Check the status bar for prompts.** The *status bar* at the bottom of the SolidWorks window describes commands as you scroll through the menus or point to buttons on toolbars. It also provides prompts for some commands that require multiple steps.



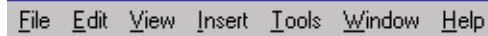
---

**NOTE:** You can hide or display the status bar. Click **View, Status Bar**. A check mark next to the menu item means the status bar is visible.

---

## Keyboard Shortcuts

Keyboard shortcut keys are available for every menu item. Look for the underlined letters in the main menu bar:

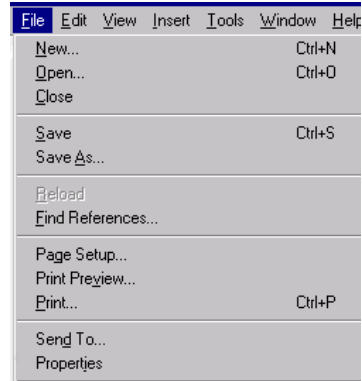


Press the **Alt** key and the underlined letter to display the menu. For example, press **Alt + F** to pull down the **File** menu.

Also, look for the underlined letter for each of the menu items. When the menu is pulled down, pressing an underlined letter activates the related command. For example, press **Alt + F** to pull down the **File** menu, then press **C** to close your file.

Some commands also have accelerator keys that are displayed on the menu beside the command. For example, the combination **Ctrl + N** opens a new file.

The following table lists the default keyboard shortcuts for rotating and viewing your model.



Action	Key Combination
Rotate the model:	
• horizontally or vertically	<b>Arrow</b> keys
• horizontally or vertically 90 degrees	<b>Shift + Arrow</b> keys
• clockwise/counterclockwise	<b>Alt + left or right Arrow</b> keys
Scroll the model	<b>Ctrl + Arrow</b> keys
Zoom in	<b>Z</b>
Zoom out	<b>z</b>
Rebuild the model	<b>Ctrl + B</b>
Force rebuild the model and all its features	<b>Ctrl + Q</b>
Redraw the screen	<b>Ctrl + R</b>

Additionally, you can customize the keyboard shortcut keys to suit your style of working.

### To customize your keyboard:

- 1 Click **Tools, Customize**.
- 2 Click the **Keyboard** tab.
- 3 From the **Categories** box, select the menu that contains the command you want.



- 4 From the **Commands** box, select the command for which you want to assign or remove a shortcut key.

The **Current Key** box displays the current shortcut key for the selected command.

The **Description** box displays the command description for the selected command.

- 5 Click **Remove** to remove the shortcut key for the selected command.

– or –

To specify a new shortcut key, type the new key or key combination in the **Press new shortcut key** box, and click **Assign**.

- 6 If you are not satisfied with the changes you made, click **Reset All** to reset the shortcut keys to their original state.
- 7 Click **OK** to accept the changes and close the dialog box.

## Preparation for Printing

---

### Page Setup

Use **File, Page Setup** to set the margins and page orientation that works best with your printer or plotter. This setting applies for all SolidWorks documents that you print, until you change the setup.

You can also use **Page Setup** to create custom headers and footers for individual documents before printing.

#### *To set print options:*

- 1 Click **File, Page Setup**.
- 2 On the **Printer** tab:
  - **Paper margins.** Click **Use printer's margins** to use the printer's default settings for margins,  
– or –  
Specify margin widths in the appropriate margin boxes (**Top, Bottom, Left, or Right**).
  - **Line weights.** Specify the line weight in the appropriate line style boxes (**Thin, Normal, and Thick** through **Thick 6**).
  - **Page orientation.** Select either **Portrait** (vertical page orientation) or **Landscape** (horizontal page orientation).
  - **Scale.**  
For **Drawings**, you can select **Scale to fit** to force the drawing onto the printer's page.  
– or –  
In the **Scale** box, click the arrows to specify the scale to use to print the drawing. (100% = 1:1)

For **Parts/Assemblies** you can select **Print Window** to print the contents of the graphics area. If you deselect **Print Window**, the model is centered on the page and printed with the **Scale** that you specify.

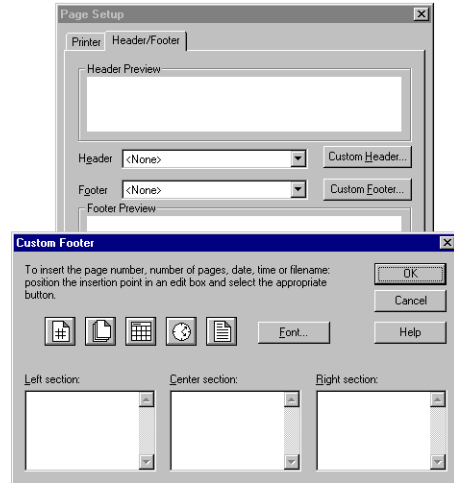
- **Print drawings in color.** Select this option to print drawings in color. (You must have a printer that is capable of printing in color and you must select **Color** on the printer's **Properties** dialog box.)
  - **Print Setup.** Sets the printer's options and provides access to the **Properties** dialog.
- 3 Click **Apply** to accept the values displayed. Click **OK** to close the dialog box.

**To customize a document header or footer:**

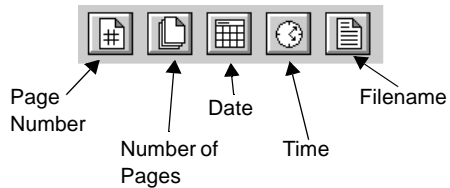
- 1 Click **File, Page Setup.**
- 2 From the **Header/Footer** tab, scroll the **Header** and **Footer** boxes to select predefined headers or footers, and view your selection in the **Preview** boxes.

– or –

Click **Custom Header** or **Custom Footer** to define your own header or footer. Click **Font** to select a font style and size.



- 3 Construct the header or footer by clicking a region (**Left section, Center section, or Right Section**), then clicking the icon that represents the kind of information to put in that region: **Page Number, Number of Pages, Date, Time, and Filename.**



- 4 Click **OK** to view the text in the preview box.
- 5 Click **Cancel** if you are not satisfied; click **Apply** to use the header or footer that you constructed.
- 6 Click **OK** to close the dialog box.
- 7 To view your changes before printing the document, click **File, Print Preview.**

## Object Linking and Embedding

---

You can use Object Linking and Embedding (OLE) to take advantage of features of other applications while in a SolidWorks document. You can also link or embed a SolidWorks part, assembly, or drawing document to another OLE-compliant application.

For example, OLE allows you to bring data generated by another application, such as a worksheet, into the SolidWorks application. Or, you may want to include a SolidWorks part in a another document, such as a product data sheet.

### Linking vs. Embedding

When using OLE, you can *link* or *embed* documents. Linking documents allows you to change the contents of a document in all places where it appears without having to edit each individual occurrence. Embedding a document allows you to keep any edits specific to the place in which you embedded it.

#### Linking Files

When you link a document, the document remains in its original location. Anything you change in the original document affects all the documents to which it is linked.

For example, if you edit a SolidWorks assembly document that is linked to several Microsoft® Word documents, the changes you make are reflected in both the original SolidWorks document and all the Word documents. Also, when you double-click on a SolidWorks object in a Word document, the SolidWorks application launches, allowing you to edit the original document (if you have the SolidWorks software installed on your computer).

---

**NOTE:** Keep track of the documents to which you link data to avoid changing information in places you do not expect.

---

#### Embedding Files

If you embed a copy of one document in another, the embedded copy becomes part of the document in which it is embedded. When you edit the embedded copy, any changes you make affect only that document. Likewise, any changes you make to the original document do not affect the embedded document.

If you embed a SolidWorks document in a Word document, and if the SolidWorks application is loaded on the computer, double-clicking the object on the Word document opens the SolidWorks application with the document active. You can edit the SolidWorks document and any changes you make affect only that document. Likewise, any changes you make to the original SolidWorks document do not affect the document embedded in the Word document.

## Using Data from Other Applications

With OLE, you can use other applications to generate data that you can then use in the SolidWorks application to control your parts, assemblies, and drawings. For example, you can calculate a dimension of a part using a math application. Or you can control the dimensions of a part using an Excel 97 worksheet. You can then link or embed the math application or worksheet to the desired dimension in the SolidWorks part document. Refer to **Design Tables** on page 8-8 for details.

### *To use data from another application in a SolidWorks document:*

- 1 With a SolidWorks document open, click **Insert, Object**.
- 2 To dynamically create and insert an object:
  - Click **Create New** in the **Insert Object** dialog box. The object appears in the SolidWorks document. Toolbars and menu options related to the object type, or application, that you selected replace the SolidWorks menus and toolbars, allowing you to use the application's tools to edit the object you inserted.

– or –

To insert an existing object:


- Click **Create from File** in the **Insert Object** dialog box. A file name appears in the **File** field.
  - Click **Browse** to select the desired object, and click **OK**.
- 3 To link the object to the SolidWorks document, click the **Link** check box in the **Insert Object** dialog box. Otherwise the object is embedded.
  - 4 To insert the object as an icon in the SolidWorks document, click **Display As Icon**.
  - 5 Click **OK** to close the **Insert Object** dialog box.

### *To change the size or scale of an OLE object in a SolidWorks document:*

- 1 Right-click the OLE object and select **Properties**.
- 2 Type a value in either the **Height**, **Width**, or **Scale** box. The values in all boxes update to remain proportional.
- 3 Click **OK** to close the box and apply the new value.

### *To delete the OLE object, do one of the following:*

- Select the object and click **Edit, Delete**.
- Select the object and press the **Delete** key.
- Right-click the object and select **Delete** from the shortcut menu. Click **Yes** at the prompt.

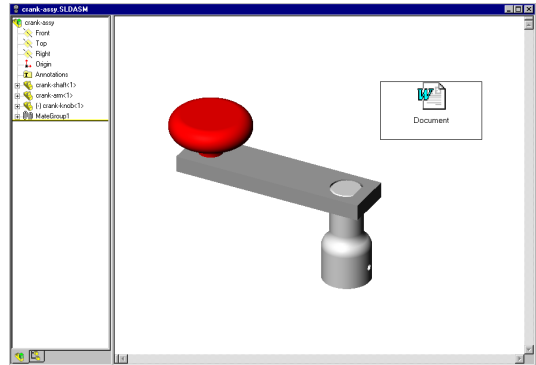
You can undo the deletion of any OLE object that is inserted into a SolidWorks document. Press **Undo** , or click **Edit, Undo Delete**.

## An OLE Object from Another Application in a SolidWorks Document

In this example, a Microsoft Word document, linked to the SolidWorks application, contains information about one of the components. You can double-click on the Microsoft Word icon to view the contents of the document.

When you link an OLE object to a SolidWorks drawing document, the default behavior is that the OLE object appears on every sheet of a multi-sheet document. If you want the OLE object to appear only on a single sheet, right-click the icon and select **Show on this sheet only** from the menu.

You can drag and drop selected OLE objects into SolidWorks documents. For example, if you drag highlighted text from a Microsoft Word document into an open SolidWorks document, the text becomes an embedded object.



## Bringing SolidWorks Data into Other Applications

Using OLE, you can link or embed the SolidWorks document with any other OLE-compliant application. For example, you can place a part in a product data sheet you created with Microsoft Word.

### **To use a SolidWorks document in another application:**

- 1 In the desired application, click **Insert, Object**.  
A dialog box appears allowing you to insert an existing object or to dynamically create and insert a new object. The dialog box that appears depends on the application you are using.
- 2 Select the desired options, including whether you want to link the SolidWorks document to this document and whether you want the SolidWorks document to appear as an icon in this document.  
The SolidWorks document or its icon appears.

## 2D Sketching and Command Emulator

---

If you are familiar with 2D drafting with AutoCAD® using a command line interface, you can continue sketching in the same way by using the SolidWorks 2D Command Emulator. This is available as a standard add-in. To activate the 2D Command Emulator, click **Tools, Add-Ins** and select **Sld2DEmu**.

By default the command insertion window is docked at the bottom of the screen, but you can move it to another position. To turn the visibility of the command line window on or off, click **2D Command Emulator** on the View menu. For more information about using the 2D Command Emulator, refer to the online help.

## Rendering Models with the PhotoWorks Application

---

PhotoWorks™ is a photo-realistic rendering application that lets you create realistic images directly from SolidWorks models. PhotoWorks is a separately purchased and licensed application that can be ordered from the dealer from whom you purchased SolidWorks. When installed, PhotoWorks is available as an add-in to the SolidWorks 99 application and is accessible by clicking **Tools, Add-Ins**.

Using the PhotoWorks application, you can specify model surface properties such as color, texture, reflectance, and transparency. PhotoWorks is supplied with a library of surface textures (metals, plastics, and so on) and, in addition, you can scan in and use your own bit-mapped surface textures, materials, scenery, and logos. You can find detailed instructions on how to use the PhotoWorks application in the *SolidWorks 99 Tutorial*, Chapter 14.

## Identifying Features with FeatureWorks

---

FeatureWorks™ is an application that recognizes a limited set of features on an imported solid in a SolidWorks part document. Once these features have been recognized, they are added to the FeatureManager design tree to be modified like to any other SolidWorks features. FeatureWorks is primarily intended for machined parts and sheet metal parts.

FeatureWorks is a separately purchased and licensed application that can be ordered from the dealer from whom you purchased the SolidWorks software. Fully integrated with SolidWorks, FeatureWorks is supplied as a SolidWorks dynamic link library (.DLL) add-in to the SolidWorks 99 application. It is accessible by clicking **Tools, Add-Ins**.

You access all the controls for the FeatureWorks interface from the FeatureWorks item on the main SolidWorks menu bar. This menu item is displayed whenever a SolidWorks part is open. For more information, see **FeatureWorks** on page 14-19.

## SolidWorks Application Programming Interface

---

The SolidWorks Application Programming Interface (API) is an OLE programming interface to SolidWorks. The API contains hundreds of functions that can be called from Visual Basic, VBA (Excel, Access, and so forth), C, C++, or SolidWorks macro files. These functions provide the programmer with direct access to SolidWorks functionality. Refer to Appendix B in this *User's Guide* for an overview of the SolidWorks API.

For a detailed description of the API and a list of all the new functionality added to the API since the last release of the SolidWorks software, refer to the API online help file. This help file, **API\_HELP.HLP**, is in the **..\SAMPLES\APPCOMM** folder of your SolidWorks installation. You can also access the API help from the Windows **Start** menu. Click **Start, Programs, SolidWorks 99, API Information**.

There is also a detailed description of the API functions on the SolidWorks Web page (<http://www.solidworks.com/html/products/api>).

For a list of the new functions added to the API for SolidWorks 99, refer to the SolidWorks online help.

## Sketching

---

Most SolidWorks features begin with a 2D sketch. Becoming comfortable with using the sketch tools is, therefore, very important. The discussion of sketching in this chapter is enough to get you started, but you should also do the “40-Minute Running Start” in the *SolidWorks 99 Tutorial*. Creating the parts in the rest of the tutorial chapters will acquaint you with many sketching techniques, as well.

You can also create 3D sketches with SolidWorks 99. In a 3D sketch, the entities exist in 3D space; they are not related to specific sketch planes. The discussion of 3D sketching begins on page 2-50.

For information about using sketches to create solid model geometry, see Chapter 4, “Creating Features.”


This chapter describes:

- Starting a new sketch
- Accessing and using the sketching tools
- Taking advantage of inferencing
- Deriving a new sketch from another
- Dimensioning the sketch and adding and deleting geometric relations
- Fully defining sketches
- Sketching on the face of a part
- Using the PropertyManager for direct editing of a sketch
- Selecting a starting plane for a sketch
- Deciding how complex the sketch should be
- 3D sketching



## Starting a New 2D Sketch

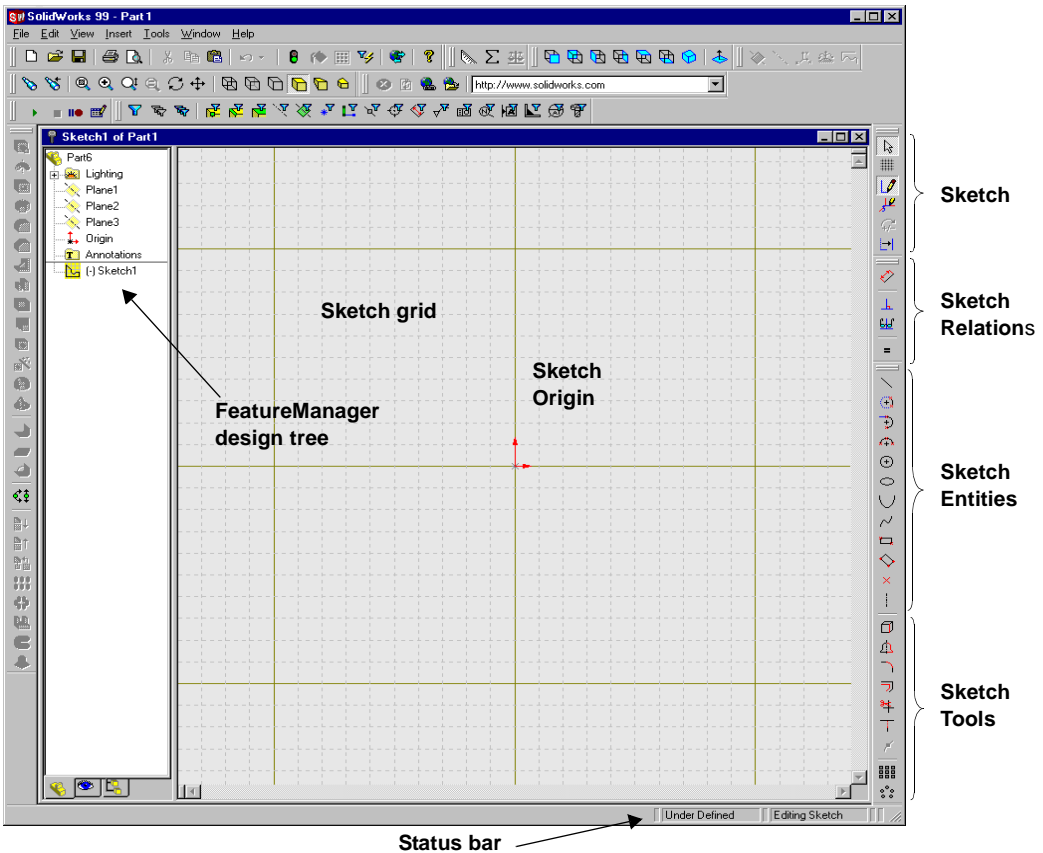
To open a new sketch, you can either

- Click the **Sketch** tool  on the Sketch toolbar, or
- Click **Insert, Sketch** on the menu bar.

A new sketch opens on Plane1 (the default plane).

- To sketch on a different plane, select the plane in the FeatureManager design tree before you open the sketch.
- To sketch on a model face, click the face before you open the sketch.

When a sketch opens, the sketch toolbars are active: *Sketch*, *Sketch Relations*, *Sketch Entities*, and *Sketch Tools*. These toolbars and tools are described on the next few pages.



For information about options that you can select to customize the sketching functionality, refer to Appendix A, “General Options” on page A-13.

To customize your toolbars by adding or removing buttons, click **Tools, Customize**. See **Customize and Arrange Toolbars** on page 1-17.

## The Sketch Toolbar

---

The tools on the Sketch toolbar act on the sketch as a whole, rather than on the individual sketch entities.



**Select** is a most versatile tool that is used throughout the SolidWorks application. Click **Tools, Select**, or right-click and choose **Select** from the shortcut menu.

In a sketch or drawing, use **Select** to

- select sketch entities
- drag sketch entities or endpoints to reshape the sketch
- select a model edge or face
- drag a selection rectangle around multiple sketch entities
- select a dimension to drag to a new location
- double-click a dimension to that you want to modify



**Grid** gives you access to grid options such as grid visibility, grid spacing, or snap behavior. Use the **Grid** button, or click **Tools, Options**, and select the **Grid/Units** tab. See **The Sketch Grid** on page 2-27.



**Sketch** opens and closes a 2D sketch. **Sketch** is on the **Insert** menu. See **Using the Sketch Entity Tools** on page 2-7 and **Using the Sketch Tools** on page 2-14.



**3D Sketch** opens and closes a sketch in 3D space. A single 3D sketch contains entities that are not related to specific sketch planes. **3D Sketch** is on the **Insert** menu. See **3D Sketching** on page 2-50.



**Modify** moves, rotates, or scales a sketch. **Modify** is on the **Tools, Sketch Tools** menu. See **Modify Sketch** on page 2-22.



**No Solve Move** lets you move sketch entities without solving the dimensions or relations in the sketch. **No Solve Move** is on the **Tools, Sketch Tools** menu. See **No Solve Move** on page 2-23.

Additionally, you can access the **No Solve Move** option when the PropertyManager functionality is enabled. See **PropertyManager Functionality** on page 2-45.

## The Sketch Relations Toolbar

---

The Sketch Relations tools have to do with dimensioning and defining the sketch entities.

The Sketch Relations toolbar and the **Tools, Dimensions** menu provide tools for dimensioning.

The Sketch Relations toolbar and the **Tools, Relations** menu provide tools for adding and deleting geometric relations. (Not all menu items have corresponding toolbar buttons.)



**Dimension** creates dimensions. The type of dimension (point-to-point, length, radial, diameter, or angular) is determined by the items you click. When you select the **Tools, Dimensions** menu, you can select the dimension type: **Parallel**, **Horizontal**, or **Vertical**. See **Dimensioning Sketches** on page 2-33.



**Add Relations** creates geometric relations (like tangent or perpendicular) between sketch entities, or between sketch entities and planes, axes, edges, curves, or vertices. See **Geometric Relations** on page 2-37 and **Add Relations Table** on page 2-39.



**Display/Delete Relations** displays relations that were either manually or automatically applied to sketch entities, and lets you delete relations that you no longer want. You can also repair entities that have errors by replacing the listed reference. See **Display/Delete Relations** on page 2-41.



**Scan Equal** displays lines and arcs with equal lengths or radii, and lets you create equal length and radii relations between the entities. See **Scan Equal** on page 2-43.

**Constrain All** sets constraints on the sketch entities of an unconstrained imported **.dxf** or **.dwg** drawing. See **Constrain All** on page 2-43.

# The Sketch Tools Toolbar

---

## The Sketch Entity tools

The sketch entity tools create individual sketch entities. You can access these tools from:

- the Sketch Tools toolbar
- the **Tools, Sketch Entity** menu
- the shortcut menu (Only those sketch tools and sketch relations tools that are appropriate at the time of the right-click are available from the shortcut menu.)

See **Using the Sketch Entity Tools** beginning on page 2-7.

Not all items on the **Tools, Sketch Entity** menu have corresponding toolbar buttons.



**Line** creates a line. See page 2-7.

The **Line** tool can also be used with 3D sketching. See page 2-50.



**Rectangle** creates a rectangle. See page 2-7.



**Parallelogram** creates a parallelogram or a rectangle at an angle to the sketch grid. See page 2-8.



**Circle** creates a circle. See page 2-8.



**Centerpoint Arc** creates an arc from a center point, start point, and endpoint. See page 2-9.



**Tangent Arc** creates an arc, tangent to a sketch entity. See page 2-9.



**3 Pt Arc** creates an arc through three points (start, end and midpoint). See page 2-9.



**Ellipse** creates an ellipse. See page 2-9.

**Centerpoint Ellipse** creates a partial ellipse from a center point, start point, and endpoint (menu item only). See page 2-10.



**Parabola** creates a parabola. See page 2-10.



**Spline** creates a spline. See page 2-10.



**Point** creates a sketched point. See page 2-12. The **Point** tool can also be used with 3D sketching. See page 2-50.



**Centerline** creates a sketched line. Used as construction geometry, and to make symmetrical sketch entities, mirror sketches, and revolved parts. See page 2-12.

The **Centerline** tool can also be used with 3D sketching. See page 2-50.

**Text** creates text in a sketch on a model face (menu item for parts only). See page 2-13.

**Construction Geometry** (only when a drawing is active) converts sketched entities (lines, arcs, splines, ellipses, and so on) in a drawing to construction geometry (menu item for drawings only). See **2D Sketching** on page 9-39. See also, **Converting Sketch Lines to Construction Geometry** on page 3-8.

## The Sketch Tools

The sketch tools act on the sketch or on individual sketch entities. You can access these tools from:

- the Sketch Tools toolbar
- the **Tools, Sketch Tools** menu

See **Using the Sketch Tools** beginning on page 2-14.

Not all items on the **Tools, Sketch Tools** menu have corresponding toolbar buttons.



**Convert Entities** creates one or more entities in a sketch by projecting an edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves onto the sketch plane. See page 2-14.



**Mirror** creates copies of sketch entities by mirroring them around a centerline. See page 2-14.



**Fillet** creates a tangent arc at the intersection of two sketch entities, trimming away the corner. See page 2-15.

The **Fillet** tool can also be used with 3D sketching. See page 2-50.



**Offset Entities** creates sketch curves offset from one or more sketch entities, model edge, loop, face, set of edges, silhouette edge, or set of external sketch curves, by a specified distance. See page 2-16.



**Trim** trims or extends a sketch entity. See page 2-16.



**Extend** extends a sketch entity. See page 2-17.



**Linear Step and Repeat** creates a linear array of sketch entities. See page 2-18.



**Circular Step and Repeat** creates a circular array of sketch entities. See page 2-19



**Split Curve** splits a curve to create two sketch entities. See page 2-20.

**Insert Spline Point** places a point on a spline to use as a handle, or to which you can add dimensions or relations. See page 2-10.

**Simplify Spline** simplifies the definition of a spline by removing spline points. See page 2-11.

**Automatic Relations** automatically creates relations as you add sketch entities. See page 2-37.

**Automatic Solve** automatically does the computation to solve the sketch geometry in the part as you create it. See page 2-21.

**Automatic Inferencing Lines** provides inferencing lines to help you work more efficiently. The dashed inferencing lines help you align endpoints, sketch lines, and model edges in a sketch or a drawing document. This functionality may impair performance, however. See page 2-29.

**Align Grid** aligns the sketch grid with the selected model edge. See page 2-21.

**Detach Segment on Drag** allows you to detach a sketch segment from other entities to which it is attached. See page 2-20.

**Override Dims on Drag** lets you override dimensions by dragging sketch entities. See page 2-21.

**Close Sketch to Model** closes an open profile sketch using existing model edges. See page 2-23.

**Check Sketch for Feature** checks sketch for errors that would prevent its use in creating the specified feature. See page 2-24.


## Using the Sketch Entity Tools

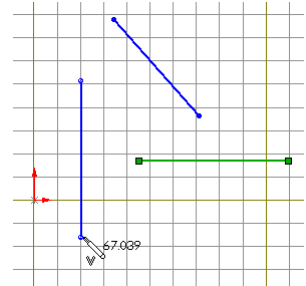
---

This section discusses how to create and use sketch entities.

### Line

#### To sketch a line:

- 1 Click the **Line** tool. 
- 2 Press the left mouse button where you want to begin sketching the line, and drag the pointer.  
A horizontal or vertical line automatically snaps to the grid points if grid snap is on.




#### To manipulate a line:

Click the **Select** tool. 


*To change the length of the line*, select one of the endpoints and drag to lengthen or shorten the line.

*To move the line*, select the line and drag the line to another position.

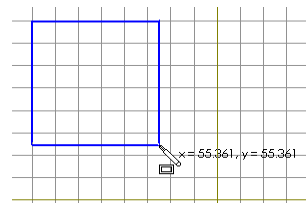
*To change the angle of a line*, select an endpoint and drag to a different angle. If the line has a vertical or horizontal relation, click **Display/Delete Relations**  and delete the vertical or horizontal relation before dragging to a new angle.

### Rectangle

#### To create a rectangle:

- 1 Click the **Rectangle** tool. 
- 2 Place the pointer where you want one corner of the rectangle to appear.
- 3 Drag the pointer and release when the rectangle is the correct size and shape.


While dragging, the dimensions of the rectangle are dynamically displayed.



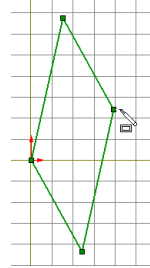
## Parallelogram

You can also use the **Parallelogram** tool to create a rectangle whose sides are not horizontal or vertical with respect to the sketch grid.


### To create a parallelogram:

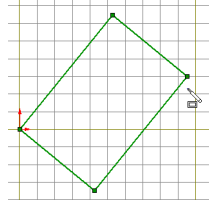
- 1 Click **Parallelogram**  on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Parallelogram**.
- 2 Place the pointer where you want one corner of the parallelogram to appear.
- 3 Drag the pointer and release when one edge of the parallelogram is the correct length.

Hold down **Ctrl**, drag the pointer, and release when the parallelogram is the correct size and shape.





### To create a rectangle at an angle:

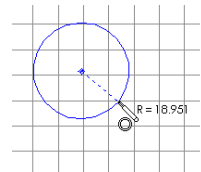
- 1 Click **Parallelogram**  on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Parallelogram**.
- 2 Place the pointer where you want one corner of the rectangle to appear.
- 3 Drag the pointer and release when one edge of the rectangle is the correct length.
- 4 Drag the pointer and release when the rectangle is the correct size and shape.



## Circle

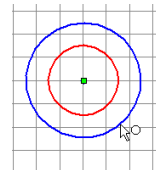
### To sketch a circle:

- 1 Click the **Circle** tool. 
- 2 Press the left mouse button where you want to place the center of the circle and drag outward to set the radius.
- 3 To manipulate the circle, click the **Select** tool. 



### To move the circle:

Point the pointer at the circle center and drag to another position.




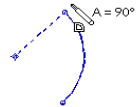
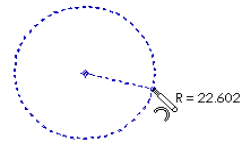
### To change the radius:

Point the pointer at the circumference and drag out or in.


## Arcs

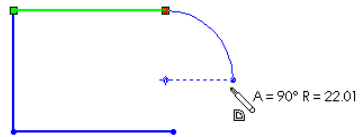
### To sketch an arc from a center point, start point, and endpoint:

- 1 Click the **Centerpoint Arc** tool. 
- 2 Press the left mouse button where you want to place the center of the arc and drag to where you want to place the start point of the arc.
- 3 Release the mouse button. The circumference guideline remains.
- 4 Press the mouse button and drag to set the length and direction of the arc, then release the mouse button.




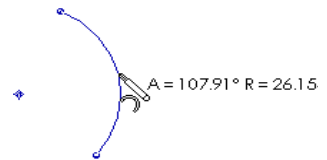
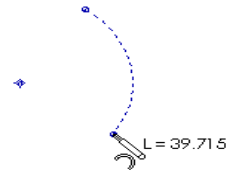
### To sketch an arc, tangent to a sketch entity:

- 1 Click the **Tangent Arc** tool. 
- 2 Press the left mouse button over the endpoint of a line, arc, or spline.
- 3 Drag out the arc to the desired shape.
- 4 Release the mouse button.




### To sketch an arc through three points:

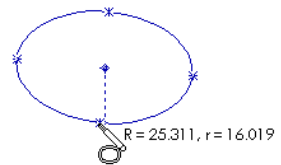
- 1 Click the **3 Point Arc** tool. 
- 2 Point where you want the arc to start.
- 3 Press the left mouse button and drag to where you want the arc to end.
- 4 Release the mouse button.
- 5 Drag the arc to set the radius and to reverse the arc, if necessary.
- 6 Release the mouse button.



## Ellipses

### To create an ellipse:

- 1 Click the **Ellipse** tool. 
- 2 Point where you want to place the center of the ellipse.
- 3 Press the left mouse button and drag to set the major axis of the ellipse.
- 4 Release the mouse button.
- 5 Press the mouse button again over the ellipse curve and drag the minor axis of the ellipse.






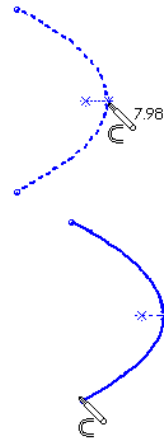
**To create a centerpoint ellipse:**

- 1 Click **Tools, Sketch Entity, Centerpoint Ellipse**.
- 2 Point to where you want to place the center of the ellipse.
- 3 Press the left mouse button, drag and release to define one axis of the ellipse.
- 4 Press the left mouse button and drag to define the second axis.
- 5 Release the mouse button. The circumference guideline remains.
- 6 Press the mouse button and drag to set the length and direction of the ellipse.


**Parabola**

**To create a parabola:**

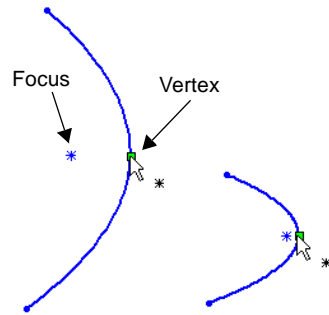
- 1 Click the **Parabola** tool. 
- 2 Point to where you want the focus of the parabola, press the left mouse button, and drag the pointer away from the focus to enlarge the parabola.
- 3 Release the button, then press it again. Drag the pointer over the section of the parabola's guide lines that you want to use.



**To shape the curve of the parabola:**


Click **Select**  and drag the vertex.

- To flatten the curve, drag the vertex away from the focus.
- To make a sharper curve, drag the vertex closer to the focus.



**Spline**

**To create a spline:**

- 1 Click the **Spline** tool. 
- 2 Press the left mouse button and drag out the first segment.
- 3 Press over the endpoint and drag out the second segment.
- 4 Repeat until the spline is complete.

---

**NOTE:** If **Alternate spline creation** is selected using **Tools, Options, General**, click on each point to create the spline, then double-click when the spline is complete.

---

### To insert a spline point:

Use inserted points to reshape a spline and to add relations or dimensions.

- 1 Right-click the spline and select **Insert Spline Point**.
- 2 Click one or more locations on the spline where you want to insert points.



You can:

- use spline points as handles to pull the spline into the shape you want
- add dimensions between spline points or between spline points and other entities
- add relations to spline points
- drag the endpoints to reshape the spline

A newly created spline is *non-proportional* by default; you can dimension and add relations to points of a non-proportional spline. You can change its shape by dragging a spline point or endpoint.

A *proportional* spline retains its shape when you drag the endpoints; the entire spline resizes proportionally. You cannot dimension or add relations to its points.

To change the proportional status of a spline, right-click the spline and select **Properties**. Click to clear the **Proportional** check box on the PropertyManager tab.

### To delete spline points:

Select a point and press the **Delete** key.

### To simplify a spline:

The **Simplify Spline** option simplifies the definition of the spline by removing through spline points. The spline is re-fitted to the tolerance of the original spline shape that you specify in the **Tolerance** box.

- 1 Right-click the spline and select **Simplify Spline** from the shortcut menu, or click the spline and then click **Tools, Sketch Tools, Simplify Spline**.

- 2 Enter a higher value in the **Tolerance** box, and click **Apply**.

The **Number of spline points** box reports the number of spline points in the original sketched spline and also reports the number of points in the simplified curve.

- 3 Observe the spline preview. If necessary to produce the spline curve that you need, reduce or increase the value in the **Tolerance** box and click **Apply** again.
- 4 When you are satisfied with the shape of the curve, and the reduced number of spline points, click **OK**.


---

**NOTE:** Use the **Simplify Spline** option to increase the performance of a model with complex spline curves.


---

## Point

### ***To create a sketch point:***

- 1 Click the **Point** tool. 
- 2 Click where you want to create the sketch point.

### ***To create a sketch point at the virtual intersection point of two entities:***

- 1 Hold down **Ctrl** and select the two entities.
- 2 Click the **Point** tool. 

A virtual sharp symbol indicates the point of intersection and retains its dimensions, even if the actual intersection no longer exists, such as when a corner is removed by a fillet.


There are several options for the appearance of the virtual sharps. To select the appearance of the point, click **Virtual Sharp** on the **Tools, Options, Detailing** page.

## Centerline

A centerline is used for construction purposes only and you cannot create geometry from it. You can use a centerline to make symmetrical sketch entities, mirror sketches, and to make revolved parts. You can dimension or add relations to a centerline.

You can convert the center line to a sketched line to use in geometry creation.

### ***To create a centerline:***

- 1 Click the **Centerline** tool. 
- 2 Press the left mouse button where you want begin sketching the centerline, and drag the pointer.
- 3 Release the mouse button.

### ***To convert a centerline to a sketched line:***

- 1 Right-click the centerline and select **Properties**.
- 2 On the PropertyManager tab, click to deselect the **For construction** check box.  
If direct edit is enabled, you just select the centerline and then click to deselect the **For construction** check box on the PropertyManager tab.

## Text

The **Text** tool creates text on a part as either an extruded boss or a cut.

---

**NOTE:** The text aligns with the x-axis of the sketch plane and proceeds in the direction of the increasing x value.

---

### **To insert text on a model face:**

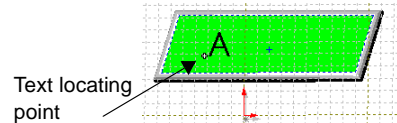
- 1 Click a model face where you want to insert text, and open a new sketch.
- 2 Click **Tools, Sketch Entity, Text**.

The pointer has a “Note” symbol attached.

- 3 Click on the model face where you want the text to begin.

The **Text** dialog box opens.



- 4 Type your text into the **Text** box.



- 5 Select the check box to use the document’s default font, or click the **Font** button and choose a font style and size. The text displays in the fonts that you select.
- 6 Click the arrows next to the **Scale** box to increase or decrease the scale of the text.
- 7 Click **OK** when you are satisfied with the appearance of the text.
- 8 You can locate the text on the face of the model by dimensioning or by adding relations to the *text locating point* in the text sketch.

– or –

You can specify the orientation and location of the text by using the **Modify Sketch** tool. (For more information, see **Modify Sketch** on page 2-22.)

- 9 Click **Insert, Boss, Extrude** or , or **Insert, Cut, Extrude** or , and set the **Depth** value.




- 10 Click **OK** to extrude the text.

### **To change text orientation and location after making the extrusion:**

- 1 Right-click the text sketch in the FeatureManager design tree and select **Edit Sketch**.
- 2 Edit the dimensions or change the relations that were specified originally,

– or –

Click **Tools, Sketch Tools, Modify Sketch** or , and use the mouse to position the text.

- 3 Exit the sketch or, click **Edit, Rebuild** or .

### **To edit the text:**

- 1 Right-click the text sketch in the FeatureManager design tree and select **Edit Sketch**.
- 2 Right-click the text and select **Properties**.
- 3 Change the text or the font and click **OK**.
- 4 Exit the sketch or, click **Edit, Rebuild** or .


## Using the Sketch Tools

---

### **Convert Entities**

The **Convert Entities** tool creates one or more curves in a sketch by projecting an edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves onto the sketch plane.

#### **To convert an entity:**

- 1 With a sketch active, click on a model edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves.
- 2 Click  or **Tools, Sketch Tools, Convert Entities**.

The following relations are created automatically:



- An **On Edge** relation is created between the new sketch curve and the entity, causing the curve to be updated if the entity changes.
- A **Fixed** relation is created internally on the endpoints of the sketch entity so the sketch can remain fully defined. This internal relation is not displayed when you use **Display/Delete Relations**. Remove the **Fixed** relation by dragging the endpoints.

### **Mirror Sketch Entities**




The **Mirror** tool creates copies of sketch entities mirrored around a centerline.

When you create mirrored entities, the software applies a *symmetric* relation between corresponding pairs of sketch segments or sketch points (the ends of mirrored lines, the centers of arcs, and so on). If you change a mirrored entity, its mirror image also changes.

#### **To mirror existing items:**

- 1 In a sketch, click the **Centerline** tool  and draw a centerline.
- 2 Hold down the **Ctrl** key and select the centerline and the items you want to mirror.
- 3 Click  or **Tools, Sketch Tools, Mirror**.

### **To mirror items as you sketch them:**


- 1 Select a centerline around which to mirror.
- 2 Click  or **Tools, Sketch Tools, Mirror**. Symmetry symbols appear at both ends of the centerline to indicate that automatic mirroring is active. 
- 3 Any entities that you sketch are mirrored automatically as you sketch them.
- 4 To turn mirroring off, click  again.

### **Fillet (Sketch)**

The **Fillet** tool creates a tangent arc at the intersection of two sketch entities, trimming away the corner.

You can also fillet non-intersecting entities, if they are not dimensioned. The selected entities are extended and then the corner is filleted away.

### **To create a fillet in a sketch:**

- 1 Hold the **Ctrl** key and select two intersecting sketch entities. You can also make your selections after the dialog box appears.
- 2 Click  or **Tools, Sketch Tools, Fillet**.  
The **Sketch Fillet** dialog box appears.
- 3 In the **Radius** box, type a fillet radius value or use the arrows to scroll to a new value.
- 4 If the corner has dimensions or relations, and you want to keep the virtual intersection point, select **Keep constrained corners**.

A virtual sharp symbol indicates the point of intersection and retains its dimensions. To change the display of the point, click **Virtual Sharp** on the **Tools, Options, Detailing** page.

- 5 If you pre-selected sketch entities, click **Apply** to create the fillet.

– or –


If you did not pre-select, click the corner that you want to fillet, or select two sketch entities that would form a corner if extended.

---

**NOTE:** Consecutive fillets with the same radius are not dimensioned individually; they have an automatic **Equal** relation with the first fillet of that radius.

---

The **Sketch Fillet** dialog box remains open so that you can continue creating fillets until you close the dialog box.

- 6 Click  to cancel the last fillet.
- 7 Click **Close** to close the dialog box.


For information about creating a fillet feature, see **Fillet/Round** on page 4-23.

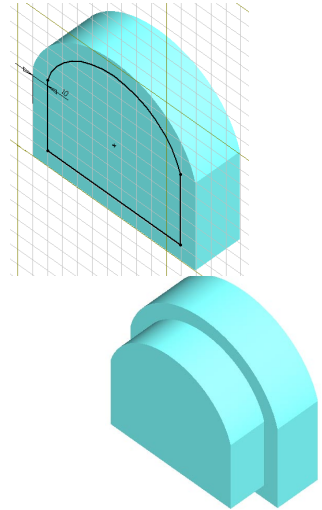
## Offset Entities

The **Offset Entities** tool creates sketch curves offset from one or more selected sketch entities, edges, loops, faces, external sketch curves, a set of edges, or set of external sketch curves by a specified distance. (The selected sketch entity must not be construction geometry, and must be either an arc, a line, or set of lines.)

An offset entity relation is created between each original entity and the corresponding sketch curve. If the entity changes when you rebuild the model, the offset curve also changes. See also **PropertyManager Functionality** on page 2-45.

### To create a sketch curve offset from a model edge:

- 1 In a sketch, select one or more sketch entities, a model face, a model edge, or external sketch curves.
- 2 Click  or **Tools, Sketch Tools, Offset Entities**.
- 3 Set the value in the **Offset** box.
- 4 If necessary, select **Reverse** to change the direction of the offset.
- 5 Click **Select Chain** to create an offset of all contiguous sketch entities.
- 6 Click **Apply**. Click **Close** to exit the dialog box.



### To change the size of the offset:

Change the amount of the offset by double-clicking the offset's dimension and changing the value.

---

**NOTE:** If the preview of the offset disappears, the offset is not valid and you must change the offset dimension or direction.


---

## Trim

Use the **Trim** tool to:

- Trim a line, arc, ellipse, circle, spline, or centerline to its intersection with a line, arc, circle, ellipse, spline, or centerline.
- Delete a line, arc, circle, ellipse, spline, or centerline.
- Extend a sketch segment until it is coincident with another entity.

### To trim a sketch entity:

- 1 Click the **Trim** tool , or **Tools, Sketch Tools, Trim**.
- 2 Move the pointer over a sketch until the sketch segment that you want to trim (delete) is highlighted in red.
- 3 Click the highlighted segment to delete it.

The segment is deleted up to its intersection with another sketch segment (line, arc, circle, ellipse, spline, or centerline).

The entire sketch segment is deleted if it does not intersect with any another sketch segment.


***To use the Trim tool to extend a sketch entity:***

Drag the endpoint of a sketch entity until it is coincident with another entity. You cannot use the **Trim** tool to extend a sketch segment that is already coincident with another entity.

## **Extend**

Use the **Extend** tool to extend a line, arc, or centerline to meet a line, arc, circle, ellipse, spline, or centerline. You can activate the **Extend** tool multiple times to extend a sketch segment until there are no other entities with which it can intersect.

***To extend a sketch entity:***

- 1 Click the **Extend** tool  or **Tools, Sketch Tools, Extend**.
- 2 Move the pointer over the sketch entity that you want to extend (a line, arc, or centerline).

The selected entity displays in red, and a green line (or arc) previews the direction in which the entity will extend.

- 3 To accept the preview line or arc, click the sketch entity.

– or –

If the preview extends in the wrong direction, move the pointer to a position on the other half of the line or arc and observe the new preview. Click the sketch entity to accept the new preview line or arc.

The sketch entity extends to intersect with the next sketch entity that is available.



## Arrays of Sketch Entities


You can pattern sketch entities to create multiple copies of the sketch entities. You can create linear or circular arrays of sketch entities.

---

**NOTE:** You cannot edit the definition of an array of sketch entities. You can create the array, but dimensions are *not* available to edit the number of instances, spacing, or angle of the array once it is created.

---


### To create a Linear Step and Repeat:

- 1 Open a sketch on a model face and sketch an item to repeat.
- 2 With the sketch entities selected, click **Linear Sketch Step and Repeat**  or **Tools, Sketch Tools, Linear Step and Repeat**.

The names of the sketch entities appear in the **Items to repeat** box if the entities are pre-selected.

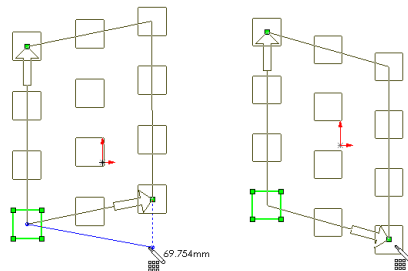
– or –

If the sketch entities are not pre-selected, click the **Items to repeat** box, then click the entities to repeat.

- 3 Under **Direction 1**:
  - In the **Number** box, set the total number of instances that you want, including the original sketch.
  - In the **Spacing** box, set the distance between instances.
  - In the **Angle** box, set an angle.
  - Click  to change the value by 180°.
- 4 Click **Preview** to see a preview of the array. After you see the preview, you can go back and adjust any of the values that you set in step 3, and click **Preview** again.
- 5 For a 2D array, repeat step 3 and insert values for **Direction 2**.


- 6 You can change the **Spacing** and **Angle** by dragging the selection point of the preview of the array.
- 7 You can delete an instance.

Notice that each instance is represented in the **Instances** box with a set of numbers that indicate the (*column, row*) position of the instance. To delete an instance in the array, select its (*column, row*) indicator in the **Instances** box and press the **Delete** key. The sketch instance disappears, and the (*column, row*) indicator moves to the **Instances deleted** box. To bring the sketch instance back, select the (*column, row*) indicator in the **Instances deleted** box, and press the **Delete** key again.



- 8 Click **OK** to complete the array of sketch entities.

### To create a Circular Step and Repeat:

- 1 Open a sketch on a model face and sketch an item to repeat.
- 2 With the sketch entities selected, click **Circular Sketch Step and Repeat**  or **Tools, Sketch Tools, Circular Step and Repeat**.


The names of the sketch entities appear in the **Items to repeat** box if the sketch is pre-selected.

– or –

If the entities are not pre-selected, click the **Items to repeat** box, then click the entities to repeat.

- 3 Under **Arc**:
  - In the **Radius** box, set a radius value. The *radius* is the distance from the center of the selected entities to the center of the array.
  - In the **Angle** box, set an angle. The *angle* is measured from the center of the selected entities to the center of the array.

- 4 Under **Step**:

- In the **Number** box, set the total number of instances that you want, including the original sketch.
- In the **Spacing** box, set the distance between instances.
- Click **Reverse Rotation**  to change the pattern rotation to counterclockwise.
- Click **Rotate** to make the instances of the selected entities rotate as they revolve around the center of the array.

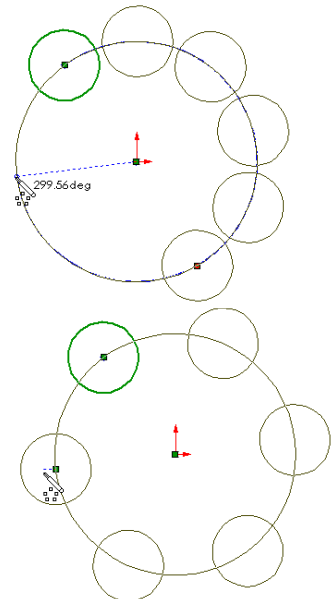
- 5 Click Preview to see a preview of the array. After you see the preview, you can go back and adjust any of the values that you set in steps 3 and 4, and click **Preview** again.

- 6 You can drag one of the selection points to set the radius and angle and the spacing between instances.

- 7 You can delete an instance.

Notice that each instance is represented in the **Instances** box with a number that indicates the position of the instance. To delete an instance in the array, select its position number in the **Instances** box and press the **Delete** key. The sketch instance disappears, and the position number moves to the **Instances deleted** box. To bring the sketch instance back, select the position number in the **Instances deleted** box, and press the **Delete** key again.

- 8 Click **OK** to complete the array of sketch entities.





## Split Curve

The **Split Curve** tool lets you split a sketch entity to create two sketch entities. You can delete a split point to combine two entities into a single entity. You must use two **Split Curve** entities to split a circle, full ellipse, or a closed spline.

You can dimension to a split point. Also, you can insert parts at split points in a piping assembly. For more information, see Chapter 13, “Piping.”

### *To split a sketch entity:*

- 1 Open the sketch containing the entity to be split.
- 2 Click **Split Curve**  on the Sketch Tools toolbar, or right-click the sketch entity, and select **Split Curve**.

The pointer changes to  when on a sketch entity that can be split.

- 3 Click the sketch entity at the location where you want the split to occur.

The sketch entity splits into two entities, and a split point is added between the two sketch entities.

### *To combine two split sketch entities into one:*

Click the split point, and press the **Delete** key.

## Detach Segment On Drag

The **Detach Segment On Drag** tool lets you detach a sketch segment (line, arc, ellipse, or spline) from the other entities it is attached to in a sketch.

---

**NOTE:** If the entity has dimensions or relations, **Detach Segment On Drag** does not delete them, so the entity may not drag or detach as you expect. Delete any relations before you use **Detach Segment On Drag**. See **Display/Delete Relations** on page 2-41.

---

### *To detach a sketch segment:*



- 1 If you have a sketch tool active, click it off (click the tool again) before trying to detach a sketch segment.
- 2 Click **Tools, Sketch Tools, Detach Segment On Drag**.
- 3 Select the entity that you want to drag away from another sketch entity and drag it to a new location.

The dragged entity detaches from any entities it shares points with.

## Override Dims on Drag

The **Override Dims on Drag** tool lets you override dimensions by dragging sketch entities. The sketch dimensions update at the end of the drag. They remain driving dimensions and update in any assemblies and drawings in which they occur.

### *To override dimensions by dragging:*

- 1 If you have a sketch tool active, click it off (click the tool again) before trying to drag a sketch entity.
- 2 Click **Tools, Sketch Tools, Override Dims on Drag**.  
When **Override Dims on Drag** is enabled, you can drag sketch entities and the dimensions will update automatically.
- 3 Click  or  to rebuild the part and exit the sketch.

## Automatic Solve

The **Automatic Solve** tool does the computation to solve the sketch geometry of your part as you create it. **Automatic Solve on** is the default.

When you are in the process of changing many dimensions in an active sketch, you may want to turn off the **Automatic Solve** capability temporarily to improve performance.

### *To turn automatic computation on or off:*

Click **Tools, Sketch Tools, Automatic Solve**. A check mark means that computations are solved automatically.

## Align Grid

The **Align Grid** tool aligns the sketch grid with the model edge that you select.

### *To align the grid to a model edge:*

- 1 Open a sketch on a model face.
- 2 Click **Tools, Sketch Tools, Align Grid**.
- 3 Click the edge to which you want to align the grid.
- 4 Click **Apply** to apply the change.

The grid shifts to align with the edge you selected, and the origin is placed at the end of the edge closest to where you clicked.

- 5 Click **OK** to accept the change and close the dialog box.

– or –


Click **Close** to close the dialog box. If you previously clicked **Apply**, the change is saved; if you did not click **Apply**, the dialog closes without change.

## Modify Sketch

The **Modify Sketch** tool  moves, rotates, or scales a sketch.

The **Translate** and **Rotate** functions move the entire sketch, including its origin. Subsequent changes are based on the updated coordinate system.

### To move or rotate a sketch:

- 1 Open a sketch or select a sketch in the FeatureManager design tree.
- 2 Click  or click **Tools, Sketch Tools, Modify**.
- 3 *To move the sketch geometry incrementally:* In the **Translate** section, type incremental values in the **X value** and **Y value** boxes, and then press **Enter**.


*To move a specified point of the sketch to a specific location:* In the **Translate** section, click **Position selected point**, and select a point on the sketch. Then type the sketch coordinates in the **X value** and **Y value boxes**, and press **Enter**.


---

**NOTE:** The **Modify Sketch** command translates the entire sketch geometry in relation to the model (including the sketch origin). The sketch geometry does not move relative to the origin of the sketch.

---

– or –

You can use the mouse-shaped pointer  to move and rotate the sketch, as follows:

- Press the right mouse button and drag to rotate the sketch around the black sketch origin symbol.
  - Press the left mouse button and drag to move the sketch.
  - Point at the endpoints or center of the black origin symbol to display one of three *flip* symbols on the right mouse button. Press the flip symbol to flip the sketch on the **X** axis, the **Y** axis, or both. 
  - Point at the center point of the black origin symbol to display a point symbol on the left mouse button. Press the left button and drag to move the center of rotation independently of the sketch.
- 4 Click **Close** to exit the dialog box.

---

**NOTE:** You cannot move a sketch if it has multiple external references. (The mouse-shaped pointer has a ? on the left button.) If the sketch has only one external point, you can rotate the sketch around this point.

---

### To scale a sketch:

- 1 In the **Scale About** box select one of the following:  
**Sketch origin** - Applies a uniform scale about the origin of the sketch.

– or –

**Moveable origin** – Scales the sketch about the moveable origin (the black origin).

- 2 Type a decimal value in the **Factor** box, and press **Enter**.
- 3 Click **Close** to exit the dialog box.

---

**NOTE:** You cannot scale a sketch if it has external references.

---


## No Solve Move

The **No Solve Move** tool lets you move sketch entities without solving the dimensions or relations in the sketch.

If dimensions or relations exist between the selected entities and other sketch entities or model geometry, a warning message appears. The message asks if you want to delete the constraints caused by the dimensions or relations:

- If you click **Yes**, the constraints are deleted, and the entities are moved.
- If you click **No**, the constraints are not deleted, and the selected entities are copied to the location to which you moved them.

To use **No Solve Move**,

- Click  on the Sketch toolbar.
- Click **Tools, Sketch Tools, No Solve Move**.
- Select **No Solve Move** on the PropertyManager tab. For more information, see **PropertyManager Functionality** on page 2-45.

## Close Sketch to Model

The **Close Sketch to Model** tool closes a sketch with an open profile, using existing model edges.

**To use an open profile sketch to extrude using existing model edges:**

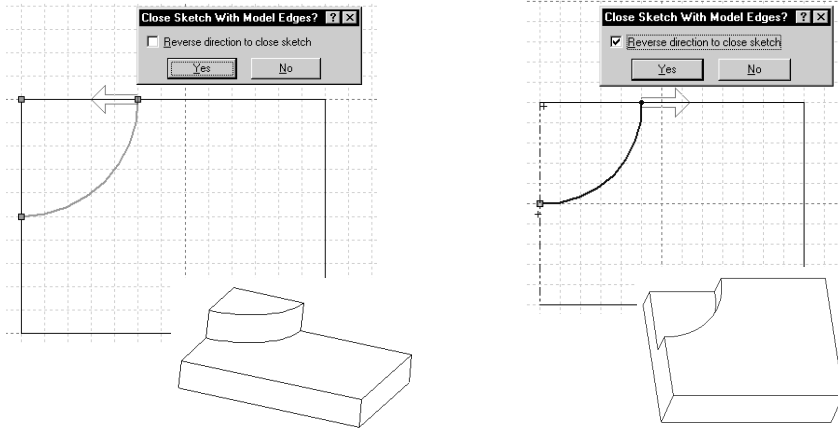
- 1 Open a sketch on a model face.
- 2 Sketch an open profile with endpoints that are coincident with the model edges on the same face boundary.
- 3 Click **Tools, Sketch Tools, Close Sketch to Model**.

The **Close Sketch with Model Edges** dialog appears.

Notice that an arrow points in the direction in which the profile will close.

- 4 Click **Reverse direction to close sketch** if it is necessary to close the sketch in the other direction.
- 5 Click **Yes** when the arrow points in the correct direction; click **No** to exit the dialog without specifying a direction.

Then you can use the closed sketch to create another feature such as an extruded boss or an extruded cut. The following examples show the effect of extruding a boss with sketches closed first in one direction and then the other.



### Check Sketch for Feature

The **Check Sketch For Feature** tool lets you check a sketch for errors that would prevent it from being used successfully in creating the type of feature you specify.

#### *To check a sketch:*

- 1 With a sketch open, click **Tools, Sketch Tools, Check Sketch For Feature**.

The **Check Sketch For Feature Usage** dialog appears.

For each **Feature usage** type, its required **Contour type** is displayed. If you select a new **Feature usage** from the list, the corresponding **Contour type** changes dynamically. Each **Contour type** forces a different set of checks on the sketch in addition to the general checks that are common to all contour types. (See the table of **Legal Contour Types** that follows.)

- If the sketch has been successfully used to create a feature, the feature type is displayed in the **Feature usage** box.
- If the sketch has not been used to create a feature, **<none>** is displayed in the **Feature Usage** box and the contour is checked for errors that are common to all contour types.

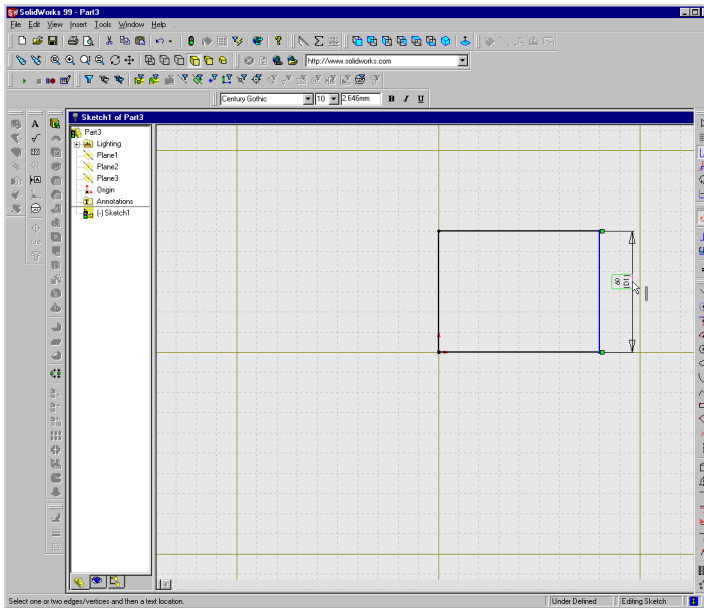
- 2 Click **Check**. If the sketch passes the checks, a “**No problems found**” message is displayed. If there is an error, a message describing the error is displayed and, if appropriate, the area of the sketch containing the error is highlighted. It reports one error for each check.
- 3 To check the sketch for possible use in other feature types, select a new **Feature usage** type from the list and click **Check**.
- 4 Click **Reset** to return to the original **Feature usage** type.
- 5 Click the **Close** button to close the dialog box. The dialog box also closes when you exit the sketch, rebuild, or change to a new document.

<b>Legal Contour types</b>	<b>Features that use this contour type</b>
General	<none> (The contour is checked for errors that are common to all contour types.)
Multiple Contained Closed (If there is more than one contour, one contour must contain the rest.)	Base Extrude
Multiple Disjoint Closed	Boss Extrude, Cut Extrude
Single Open or Closed	Loft, Loft Section, Surface Loft Section, Loft Guide, Sweep Path or Guide, Surface Sweep Section, Cut Extrude Thin, Surface Revolve, Boss Revolve Thin, Base Revolve Thin, Surface Extrude, Boss Extrude Thin, Base Extrude Thin
Single Open	Rib, Cut Extrude
Single Closed	Base Revolve, Sweep Section, Cut Revolve Thin, Cut Revolve, Boss Revolve



## Working in a Sketch

When a sketch is active, the sketch window displays the sketching toolbars, the sketch grid, as well as helpful information in the status bar.



The following information is in the status bar at the bottom of the sketch window.

- The coordinates of the pointer's location when sketching an entity.
- The state of the sketch: Over Defined, Under Defined, or Fully Defined. (See **Fully Defining Sketches** on page 2-44 for more information.)
- The text, "Editing Sketch." This makes it obvious that you are in sketch mode if you choose to turn off the sketch grid while you are working.
- When the pointer passes over a menu item or tool button, look for a brief description of the menu item or button on the left side of the status bar.

By default a sketch opens on Plane1, but you can open a sketch on Plane2 or Plane3.

### **To sketch on Plane2 or Plane3:**

- 1 Click **Plane2** or **Plane3** in the FeatureManager design tree.
- 2 Click **View, Orientation**, and double-click **\*Normal To**.
- 3 Open a sketch.

To insert an alternate plane, see **Creating a Construction Plane** on page 3-2.

## The Sketch Origin

The sketch origin is displayed in red in the active sketch. Use the sketch origin to help you understand the coordinates of the sketch that you are working on.


Every sketch in the part has its own origin, so there usually are multiple sketch origins in a part. When a sketch is open, you cannot turn off the display of its origin. You cannot dimension to, or add relations to a sketch origin.

Keep in mind that the sketch origin and the origin of the part, which displays in grey, are not one and the same.

## The Sketch Grid


By default, the SolidWorks sketching window provides a grid on which you can sketch. The grid guides you as you create your sketch and is generally a helpful tool. The grid is optional, however, and you may sketch without a grid.

### To turn the grid off:

- 1 Click the **Grid** tool , or click **Tools, Options** and select the **Grid/Units** tab.
- 2 Click to clear the **Display grid** check box.

The sketching window also provides a **Snap to points** option to help you create accurate lines and points. When grid snap is on, points that you sketch or drag snap to the nearest intersection of the grid lines. This applies to the endpoints of lines and the center points of circles and arcs.

### To turn grid snap off:

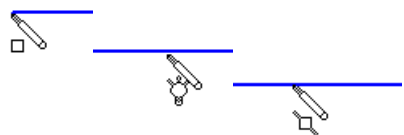
- 1 Click the **Grid** tool , or click **Tools, Options** and select the **Grid/Units** tab.
- 2 Under **Snap behavior**, click to clear the **Snap to points** check box.

For more information about additional ways to customize the sketch grid, such as changing the grid spacing, see “Grid/Units Options” on page A-16.

## Inferencing Pointers and Lines

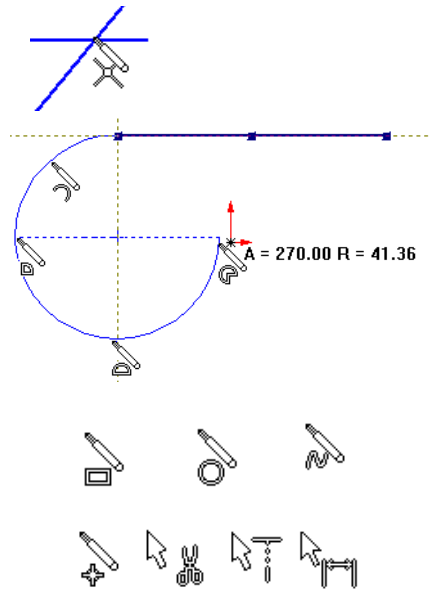
While sketching, notice that the appearance of the pointer changes to provide feedback about the pointer’s current task, position, and the geometric relations that are automatically applied.

- When you move the pointer along the length of any sketched curve, it first appears as an *endpoint* pointer, changes to the *on-curve* pointer, and then to a *midpoint* pointer, and so on.



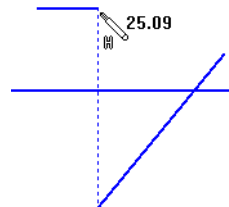
## Chapter 2 Sketching

- ❑ When two lines intersect, the pointer changes to indicate the intersection.
- ❑ When you sketch an arc, the pointer changes as it moves around the arc.
- ❑ The pointer provides information about dimensions as you sketch lines or arcs, such as the length, angle or radius of the sketch entity.
- ❑ When you select the various sketch or dimension tools, the pointer carries an appropriate symbol with it. Shown are the **Rectangle**, **Circle**, **Spline**, **Point**, **Trim**, **Extend**, and **Dimension** pointers, but there are many others.



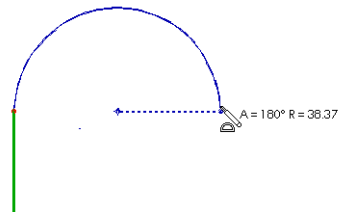
As you sketch, the SolidWorks software monitors your activity and provides *inferencing lines* to help you work more efficiently. While sketching, notice that dashed inferencing lines align the pointer with lines or points that you have already sketched and with existing model geometry.

- ❑ When the endpoint of any line you are creating aligns with another point that you have already sketched, a dashed inferencing line indicates this alignment. This helps you align endpoints with each other so you can sketch without using a grid.



Brown inference lines indicate that a relation is added automatically; blue inference lines indicate that no relation is added.

- ❑ When you sketch arcs, inferencing lines and dimensions guide you to create the size and shape arc that you want.



## Automatic Inferencing Lines

You can turn off automatic inferencing lines while editing a sketch or while sketching in a drawing document. If automatic inferencing lines are turned off, you no longer automatically infer to all model edges, active sketch entities, or non-active sketch entities.

Turning off inferencing lines in a complex sketch or drawing may improve sketching performance and reduce the chance of unwanted relations.

### ***To turn off the automatic inferencing lines:***

Click **Tools, Sketch Tools, Automatic Inferencing Lines** to turn off the automatic inferencing lines.

In drawings, you must activate the inferencing lines for current sketch entities. You have to activate the inferencing lines for each new entity that you sketch.

### ***To activate the inferencing lines for an entity in a drawing:***

- 1 Click a sketch entity tool, such as **Line** or **Circle**.
- 2 Move the pointer over the entity you want to infer to.  
The pointer changes color once the inferencing lines are active for the entity.
- 3 Sketch the new entity with inferencing lines from the activated entity.

## Using Silhouette Edges

You can select the silhouette edge of a rounded surface, such as a cylinder, revolve, loft, and sweep, to do any of the following operations: **Convert Entities**, **Offset Entities**, and add both **Dimensions** and **Relations**.

### ***To use a silhouette edge:***

- 1 Working with a sketch open on a model face or plane (or working in an active drawing view), move your pointer over a cylindrical face until the pointer changes to include a small cylinder next to the pointing arrow. The cylinder's silhouette edge becomes visible as a dashed line.



---

**NOTE:** Silhouette edge visibility is an option that you set by clicking **Tools, Options, Edges**, and selecting **Dynamic highlight from graphics view**.

---

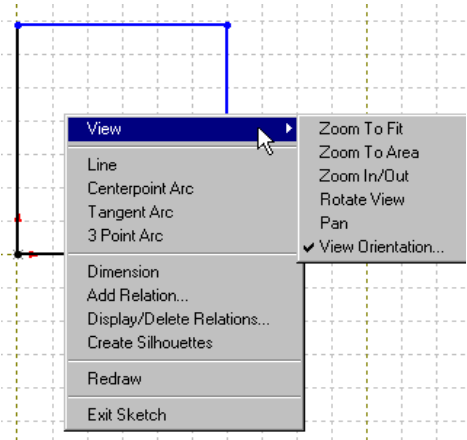
- 2 Select the dashed line to use with the **Convert Entities** or **Offset Entities** sketch tools, or to add **Dimensions** and **Relations** to your sketch.

## The Shortcut Menu

As you become more proficient in using the sketch tools, you will find that the shortcut menu provides a quick and easy way to change tasks.

If you right-click after you sketch an entity, the shortcut menu lets you select a different sketch tool without moving the pointer to the toolbar or main menu.

The shortcut menu changes depending on the task that you are performing. Only those actions that are appropriate are listed as menu items.



## Cutting, Copying, and Pasting Multiple Sketch Entities

You can cut and paste, or copy and paste, one or more sketch entities from one sketch to another, or within the same sketch, by dragging and dropping.

To copy multiple entities, press the **Ctrl** key while selecting them, then drag while using the following modifier keys:

- To *copy* within the document or to another document, hold down the **Ctrl** key while dragging.
- To *move* (cut and paste) within the same sketch, hold down the **Shift** key.
- To *move between different documents*, hold down the **Ctrl** key and drag the sketch into the second document. Release the **Ctrl** key, and drop the sketch.

Additionally, you can copy by selecting one or more sketch entities and clicking **Edit, Copy**, or by pressing **Ctrl+C**. Then you can paste the entities by clicking **Edit, Paste**, or by pressing **Ctrl+V**.

## Copying and Pasting Entire Sketches


You can copy an entire sketch and paste it on a face in the current part or you can paste it into a different sketch, or part, assembly, or drawing document.

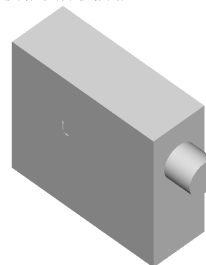
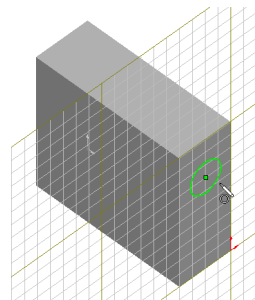
### ***To copy and paste a sketch:***

- 1 Select the sketch in the FeatureManager design tree.
- 2 Click **Edit, Copy**, or press **Ctrl+C**.
- 3 Click in the sketch or document where you want to paste the sketch.
- 4 Click **Edit, Paste**, or press **Ctrl+V**.

## Creating a Sketch on the Face of a Part

### *To sketch on model faces:*

- 1 Click the planar model face on which you want to sketch.
- 2 Click  or click **Insert, Sketch** from the main menu.
- 3 The following happens:
  - A grid is displayed on the face that you selected.
  - The sketch toolbar is active.
  - The message “Editing Sketch” appears on the status line at the bottom of the window.



To sketch on a different face, you need to click the new face and open a new sketch.

When you are editing a sketch in **Shaded** mode, you have the option to display the sketch plane as a translucent plane.

### *To turn the translucent sketch plane on or off:*

- 1 Click **Tools, Options, General** and select **Display plane when shaded**.  
If the display is slow, due to the translucent plane, it may be because of the **Transparency Quality** option specified in **Tools, Options, Performance**. With some graphics cards, the display speed will improve if you use **Low (Screen Door)** transparency.
- 2 Click **Tools, Options, Performance** and select **Low (Screen Door)** transparency if necessary to improve the speed of the display.

## Editing a Sketch



To open a sketch for editing, either select the sketch from the graphics area or the FeatureManager design tree. You can edit a sketch by adding, deleting, and dragging sketch entities, changing dimensions, changing or adding relations, and so on.

### *To open a sketch for editing using the shortcut menu:*

- 1 Right-click one of the following:
  - a face of a single-sketch feature (not a loft, for example)
  - a sketch entity on a part (when the sketch is not active)
  - a sketch name in the FeatureManager design tree.
- 2 Select **Edit Sketch** from the shortcut menu.  
The sketch used to create the selected face or entity opens for editing.

## Exiting a Sketch

You exit a sketch when you:

- Create a feature from your sketched profile. For example, when you extrude a base, boss, or a cut from your sketch.
- Click the **Rebuild** tool  when you are in a sketch.
- Click the **Sketch** tool  when in a sketch.
- Click **Insert, Sketch** from the main menu.
- Select **Exit Sketch** from the shortcut menu.

## Edit Sketch Planes

You can change the plane of a sketch to change the location of a feature on a model. This is also useful when you are working with an imported sketch.

### *To change the plane of a sketch:*

- 1 Right-click the sketch in the FeatureManager design tree, and select **Edit Sketch Plane**.
- 2 Select a new plane by clicking a plane in the FeatureManager design tree or select a new face by clicking a different planar model face.
- 3 Click **Apply** or click **Cancel** to exit without making a change.

## Derived Sketch

You can derive a sketch from another sketch that belongs to the same part or a sketch from a different part in the same assembly.

Note the following:

- If you delete a sketch from which a new sketch was derived, you are prompted that all derived sketches will automatically be underived. Click **Yes** or **No**.
- You cannot add or delete geometry in a derived sketch; its shape is always the same as its parent. However, you can re-orient it using dimensions and geometric relations.
- When you make changes to the original sketch, the derived sketch updates automatically.
- To break the link between the derived sketch and its parent sketch, right-click the name of the derived sketch or part in the FeatureManager design tree and select **Underive** from the menu. (After the link is broken, the derived sketch no longer updates when a change is made to the original sketch.)

**To derive a sketch:**

- 1 Select the sketch from which you want to derive a new sketch.
- 2 Hold down the **Ctrl** key and click the face on which you want to place the new sketch.
- 3 Click **Insert, Derived Sketch**.

The sketch appears on the plane of the selected face, and the sketch is opened for editing.

- 4 Position the derived sketch by dragging and dimensioning it to the selected face. (The derived sketch is rigid and drags as a single entity.)

**To derive a sketch from a sketch in the same assembly:**

- 1 Right-click the part in which you want to place a derived sketch.
- 2 Select **Edit Part**.
- 3 Select the sketch (in the same assembly) from which you want to derive a new sketch.
- 4 Hold down the **Ctrl** key and click the face on which you want to place the new sketch.
- 5 Click **Insert, Derived Sketch**.
- 6 Position the derived sketch by dragging and dimensioning it to the selected face. (The derived sketch is rigid and drags as a single entity.)

## Adding Dimensions and Relations

---

### Dimensioning Sketches


Usually, you dimension your sketch as you go along. However, when using the SolidWorks application, it is not *necessary* to dimension sketches before you use them to create features. When you add or change dimensions in a sketch, the changes are reflected in the part and the drawing views of the part.

---

**NOTE:** This section describes dimensioning sketches in the graphics area. If you are more familiar with 2D drafting applications that use an alternate way of dimensioning sketch entities, you may want to use The PropertyManager functionality described on page 2-45.


---

There are two methods of dimensioning a sketch directly in the graphics area:

- You can select the dimension tool for the specific type of dimension you want.
  - Click , and right-click to select either **Horizontal Dimension**, or **Vertical Dimension** (the default dimension is **Parallel**).
  - or –
  - Click **Tools, Dimensions** and select either **Parallel**, **Horizontal**, or **Vertical**.




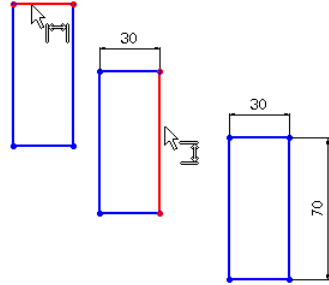
## Chapter 2 Sketching

- ❑ You can click  and preview the dimension before placing it.


For linear dimensions, the preview automatically switches between horizontal, vertical, and slanted, depending on the pointer location. For diameter dimensions, the preview switches between a radial dimension and a linear dimension. The behavior is similar for angular dimensions, where the dimension created depends on the quadrant location of the pointer.

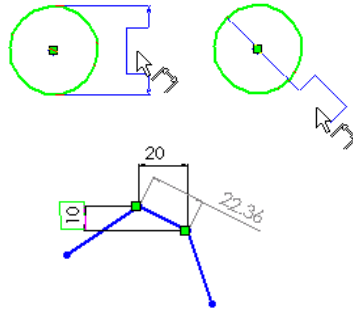
### To add a specific type dimension to a sketch:

- 1 Click  and right-click to select either **Horizontal** or **Vertical**, or click **Tools, Dimensions** and select either **Parallel, Horizontal**, or **Vertical**.
- 2 Click one or more sketch entities or model items, then click where you want to place the dimension.

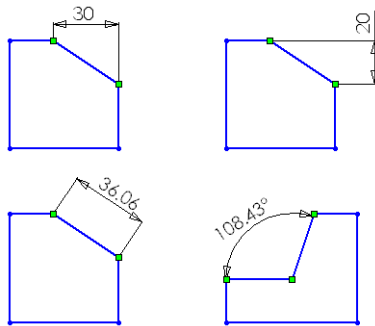


### To preview a dimension:

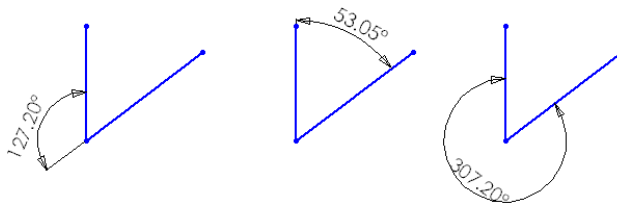
- 1 Click  and click the sketch entity, or entities, to dimension.
- 2 Move the pointer around and observe the dimension preview.
- 3 Click when the preview displays the dimension that you want.



### Examples of point-to-point dimensions:



### Examples of angular dimensions



## Dimensioning with Equations

You can insert dimensions in a sketch using mathematical relations between dimensions or parameters, and using dimension names as variables. Equations can also be used outside of a sketch.

For a more complete discussion about using and editing equations to control dimensions, see **Equations** on page 5-9.

---

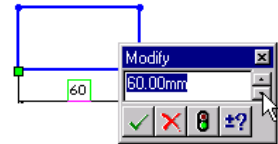
**NOTE:** Dimensions that are driven by equations cannot be changed by editing the dimension value in the model.

---

## Changing Dimensions

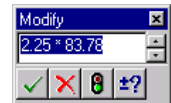
### *To change a dimension:*

- 1 Double-click the dimension.  
The **Modify** dialog box appears.
- 2 To change the dimension value, click the arrows to change the value and press **Enter**, or type a new value or a calculation and press **Enter**.



In addition, you can use the row of buttons below the value window to do any of the following:

- Accept the current value and exit.
- Restore the original value and exit.
- Rebuild the sketch or model with the current value.
- Change the spin box increment value.



---

**NOTE:** You can set the **Modify** dialog box to appear automatically every time you place a dimension. Click **Tools, Options, General** and select **Input dimension value**.

---

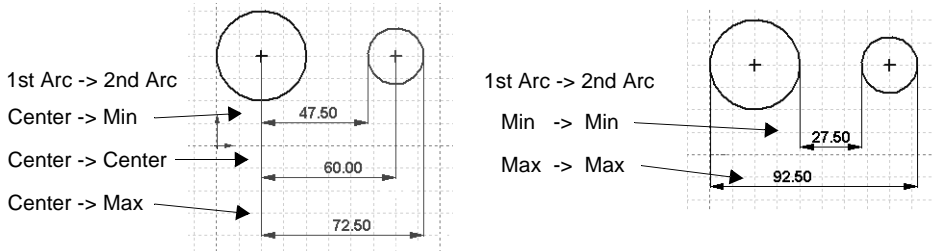
## Dimensioning Arcs and Circles

By default, when you dimension to an arc or circle, the dimension goes to the center point of the arc or circle. You can edit the dimension so that it measures to the edge of an arc or circle.


### To change dimension options for arcs and circles:

- 1 Right-click the dimension, and select **Properties** from the menu.
- 2 From the **DIMENSION PROPERTIES** dialog box, select the **First arc condition** and the **Second arc condition** that provides the dimension that you want.

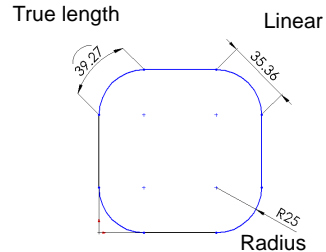
The following illustrations show some of the options for dimensioning between arcs and circles.



### To dimension the true length of an arc:

- 1 Click  or **Tools, Dimensions**.
- 2 Click the arc.
- 3 Click both endpoints of the arc.
- 4 Click to place the dimension.

An arc length symbol is added above the dimension text.




## Dimension Properties

You set the default properties for dimensions on the **Detailing** tab of the **Options** dialog box. These properties are related to the appearance of the dimensions on the drawing or a print of the model. Dimension properties include font size and style, arrow placement and style, the precision and units of measure used. See Appendix A, “Detailing Options” on page A-4.

You can modify the properties of selected dimensions at any time. See **Dimension Properties** on page 10-12 of the “Detailing” chapter for more information.

When you select **Enable direct edit of sketch**, using the **Tools, Options, General** tab, you can view and edit dimension properties of selected sketch entities on the **PropertyManager** tab. See **PropertyManager Functionality** on page 2-45.

## Geometric Relations

You can use the **Add Relations**  tool to create geometric relations (like tangent or perpendicular) between sketch entities, or between sketch entities and planes, axes, edges, or vertices.

When creating relations, at least one of the items must be a sketch entity. The other item (or items) can be either a sketch entity or an edge, face, vertex, origin, plane, axis, or sketch curve from another sketch that forms a line or arc when projected on the sketch plane.

Geometric relations can be created two ways:

- Automatically created by the SolidWorks software
- Manually created by the user

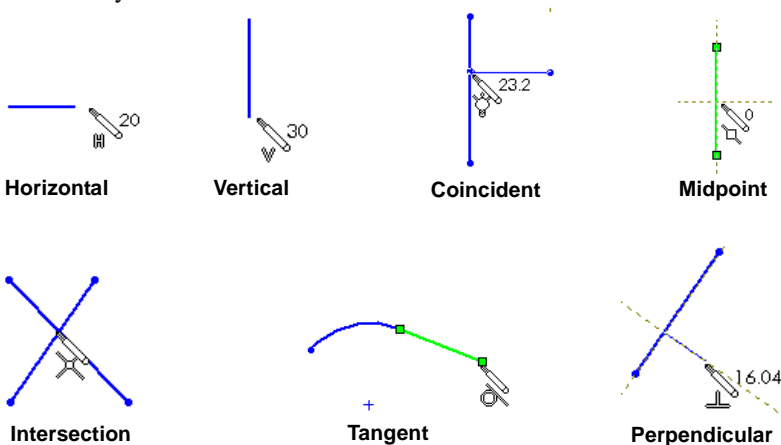
For a complete discussion of all the sketch relations, refer to the **Add Relations Table** on page 2-39.

### Automatic Relations

Turn **Automatic Relations** on or off for the current document by clicking **Tools, Sketch Tools, Automatic Relations**. A check mark next to the **Automatic Relations** menu item indicates that relations are created automatically as you sketch. For instance, if you draw a vertical line, the SolidWorks software applies a “vertical” relation to it.


To specify **Automatic Relations** for all documents, click **Tools, Options, General**. In the **Sketch** section, select **Automatic relations**.

As you sketch, the pointer changes shape to indicate which relations can be created. Note that the pointer has a small **H** when the line is horizontal; the pointer has a **V** when the line is vertical. If the **Automatic Relations** option is on, the following relations are created automatically.



## Manually Created Relations

### To apply relations manually:


- 1 Click , or click **Tools, Relations, Add**.
- 2 In a sketch, select the items (sketch entities, vertices, edges, and so forth) to relate.

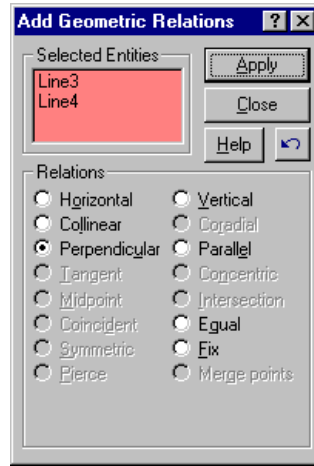
The **Add Relations** dialog box selects the best default relation based on the entities you selected, and indicates that selection with a dot in the middle of the radio button. Relations that already exist are marked with a dark circle around the button.

Relations that are not appropriate for the items you selected are “grayed out.”

For many relations, you can select more than two entities. At least one of the items must be a sketch entity. Other items can be a sketch entity, an edge, face, vertex, origin, plane, axis, or sketch curve from another sketch that forms a line or arc when projected on the sketch plane.

The items that you select are displayed in the **Selected Entities** box. To remove an item from the list, click it again in the graphics area. To remove all items, right-click in the graphics area and select **Clear Selections**.

- 3 Select the relation you want to add, then click **Apply**.  
To remove the last relation you added, click the **Undo** button .
- 4 Click **Close** to close the **Add Relations** dialog box.



## Add Relations Table

Available relations depend on the items you select. Please see the definitions of terms at the end of this table.\*

To add this relation:	Select:	Result:
<b>Horizontal or Vertical</b>	One or more lines or two or more points	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
<b>Collinear</b>	Two or more lines	The items lie on the same infinite line.
<b>Conradial</b>	Two or more arcs	The items share the same center point and radius.
<b>Perpendicular</b>	Two lines	The two items are perpendicular to each other.
<b>Parallel</b>	Two or more lines	The items remain parallel.
<b>Tangent</b>	An arc, spline, or ellipse and another arc, or line	The two items remain tangent.
<b>Concentric</b>	Two or more arcs, or a point and an arc	The arcs share the same center point.
<b>Midpoint</b>	A point and a line	The point remains at the midpoint of the line.
<b>Intersection</b>	Two lines and one point	The point remains at the intersection of the lines.
<b>Coincident</b>	A point and a line, arc, or ellipse	The point lies on the line, arc, or ellipse.
<b>Equal</b>	Two or more lines, or two or more arcs	The line lengths or radii remain equal.
<b>Symmetric</b>	A centerline and two points, lines, arcs, or ellipses	The items remain equidistant from the centerline, on a line perpendicular to the centerline.
<b>Fix</b>	Any item	The item's size and location are fixed. The endpoints are still free to move along the underlying infinite line. The endpoints of an arc or elliptical segment are free to move along the underlying circle or ellipse.

*Table continued on next page*

<b>Pierce</b>	A sketch point and an axis, edge, line, or spline	The sketch point is constructed coincident to where the axis, edge, or line pierces the sketch plane.
<b>Merge Points</b>	Two sketch points or endpoints	The two points are merged into a single point.

---

When you create a relation to a line, the relation is to the infinite line, not just the sketched line segment or the physical edge. As a result, some items may not physically touch when you expect them to.


Similarly, when you create a relation to an arc segment or elliptical segment the relation is to the *full* circle or ellipse.

Also, if you create a relation to an item that does not lie on the sketch plane, the resulting relation applies to the *projection* of that item as it appears on the sketch plane.

***\*Definitions of terms used in the Add Relations table:***

<b>Arc</b>	Can be an arc or circle in the sketch, or an external entity that projects as an arc or circle in the sketch.
<b>Ellipse</b>	Can be an ellipse in the sketch, or an external entity that projects as an ellipse in the sketch.
<b>Line</b>	Can be a line or centerline in the sketch, or an external entity (edge, plane, axis, or sketch curve in an external sketch) that projects as a line in the sketch.
<b>Point</b>	Can be a point in the sketch, or an external entity (origin, vertex, axis, or point in an external sketch) that projects as a point in the sketch.
<b>Spline</b>	Can be a spline in the sketch, or an external entity that projects as a spline in the sketch.

## Display/Delete Relations

You can view the relations on each entity by selecting the entity and clicking  or **Tools, Relations, Display/Delete**. If your sketch becomes “over defined” or “not solved,” use **Display/Delete** to examine and delete relations as necessary. (For more information about the sketch states, see **Fully Defining Sketches** on page 2-44.)



### To view or delete relations:


- 1 Click the sketch entity about which you have a question. (If no entity is selected, all relations in the sketch are listed.)
- 2 Click **Display/Delete Relations**.

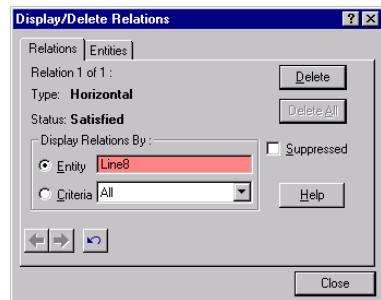
A dialog box appears that has two tabs:

- **Relations** tells what relations apply and allows you to view or delete the relations. (See **Using the Relations tab** on page 2-41.)
- **Entities** displays information about the entities involved in the active relation and lets you replace them, if necessary. (See **Using the Entities tab** on page 2-42.)

### Using the Relations tab

- 1 To specify the scope of the relations display, in the **Display Relations By** box select one of the following:
  - **Entity**, if you want to display relations for the selected entity only.
  - **Criteria**, if you want to display all the relations in the entire sketch that meet the criteria you select from the list. For example, you can choose **Dangling** to list all the dangling relations in the sketch.
- 2 Click **Previous** , or **Next** , to view each relation in turn (if there are more than one).

As each relation is displayed, the related sketch entities are highlighted, and the status is displayed. (The status can be **Dangling**, **Satisfied**, **Over Defining**, or **Not Solved**.)
- 3 Select **Suppressed** to suppress or unsuppress the current relation.
- 4 Click **Delete** to delete the current relation; click **Delete All** to delete all of the relations currently selected.
- 5 To undo an action that you just performed, click **Undo** .
- 6 You can continue to display relations for any or all entities in the sketch. The dialog box remains open until you click the **Close** button.

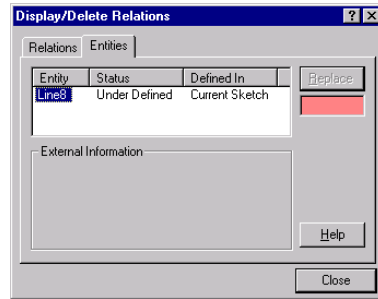





### Using the Entities tab

1 Click the **Entities** tab.

- The list box displays the name of each entity, the entity's status, and the place where it is defined: **Current Sketch**, **Same Model**, or **External Model**.
- If the entity is defined as existing in the **Same Model**, the **External Information** section lists the external **Entity** to which the selected sketch entity has a relation.
- If the entity is defined as existing in an **External Model**, the **External Information** section lists the **Assembly** name and the **Owner** part within the assembly.




2 To replace an entity:

- a) Select an entity that you want to replace in the dialog box.
- b) Click an entity in the graphics area that you want to use as a replacement. This entity's name appears in the selection box under the **Replace** button.
- c) Click the **Replace** button. The software attempts to replace the entity and solve the sketch.
- d) If it is necessary to undo the replacement, click the **Undo** button .

3 Click the **Close** button to close the dialog box.

### Reattaching Dangling Relations

When there is a single dangling relation, a red handle is displayed at the midpoint of the dangling segment. To reattach the relations, drag the handle onto a sketch or model entity that is suitable to resolve the dangling relation. The pointer displays a  symbol if the entity is unsuitable.

You must use the **Display/Delete** relations dialog box to reattach relations in the following cases:

- if there are multiple dangling relations
  - or –
- if the entity is a sketch point.

## Scan Equal

Click **Tools, Relations, Scan Equal** to scan a sketch for lines and arcs with equal lengths or radii. **Scan Equal** also provides a way to set an **Equal** relation between sketch entities that are the same length or radius.

### *To locate equal radii or line lengths:*

- 1 In a sketch, click **Tools, Relations, Scan Equal**.  
If sketch entities exist that are equal, the appropriate radio buttons are active. (For example, if two or more lines are equal, **Length** is active; if two or more arcs have equal radii, **Radii** is active; if an arc has a radius equal to a line length, **Both** is active.)
- 2 To see the equal sketch entities, click an active radio button.
  - The equal entities corresponding to the radio button that you clicked are highlighted in the sketch.
  - The **Value** box displays the length or radii.
  - The **Line Count** or the **Arc Count** box displays the number of equal entities.
- 3 If there are additional sets of equal entities of the same type, the **Find Next** button is active. To highlight the additional sets, click the **Find Next** button.
- 4 Click **Set Equal** if you want to create an **Equal** relation between the highlighted entities.
- 5 Click **Close**.

## Constrain All

The **Constrain All** tool sets constraints on the sketch entities of an unconstrained imported .dxf or .dwg drawing.

### *To solve relations in an imported drawing:*

- 1 Import a sketch from a .dxf or .dwg drawing.
- 2 Click **Tools, Relations, Constrain All**.

The command tries to solve all the apparent relations in the sketch and reports the number of constraints that were added to the sketch.

## Fully Defining Sketches

While you are creating or editing a sketch, you should be aware that a sketch may be in any of five states described below. The state of the sketch as a whole is displayed in the status bar at the bottom of the SolidWorks window.

- Fully Defined* – black. All the lines and curves in the sketch, and their positions, are described by either dimensions or relations, or both.
- Over Defined* – red. The sketch has too many dimensions or relations. For information about viewing and removing conflicting relations, refer to **Display/Delete Relations** on page 2-41.
- Under Defined* – blue. Some of the dimensions, relations, or both, are not defined and are free to change. You can drag the endpoints, lines, or curves with the pointer until the sketch entity changes shape.
- No Solution Found* – pink. The sketch was not solved. The geometry, relations, and dimensions that prevent the solution of the sketch are displayed.
- Invalid Solution Found* – yellow. The sketch was solved but will result in invalid geometry, such as a zero length line, zero radius arc, or self-intersecting spline.

With SolidWorks 99 it is *not necessary* to fully dimension or define your sketches before you use them to create features. However, it is usually a good idea to fully define your sketches.

---

**NOTE:** If you wish to always use fully defined sketches, click **Tools**, **Options**, and select the **General** tab. In the **Sketch** section, click **Use fully defined sketches**.

---

### ***Sketch Geometry may have any of the following states:***

*Fully Defined* – black. The dimensions and relations are fully and correctly described.

*Under Defined* – blue. The dimensions and relations are not defined adequately and lines may move or change size unexpectedly.

*Over Defined* – red. This geometry is constrained by too many dimensions or relations.

*Dangling* – brown, dashed. Only applies to a sketch entity that was added automatically to the sketch in the last known position of the dangling model geometry.

*Not Solved* – pink. This geometry's position cannot be determined using the current constraints.

*Invalid* – yellow. This geometry would be geometrically invalid if the sketch were solved.

**Sketch Dimensions or Relations may have any of the following states:**

*Dangling* – brown, dashed. A dimension that can no longer be resolved (a dimension to a deleted entity, for example).

*Satisfied* – black. A dimension that is completely and appropriately defined.

*Over Defining* – red. The dimension or relation is involved in over defining one or more entities.


*Not Solved* – pink. The dimension or relation results in not being able to determine the position of one or more sketch entities.

*Driven* – gray. The dimension's value is driven by solving the sketch.

## PropertyManager Functionality

---

The PropertyManager functionality provides an alternate way of using the SolidWorks 99 application that may be more comfortable for people who are familiar with 2D drafting applications. Using the PropertyManager tab, you can select a sketch entity and then directly edit values for size, coordinates, angle, and so on of a sketch entity without adding dimensions to the sketch.

To use the PropertyManager tab, click **Tools, Options, General** and select the **Enable direct edit of sketch** option. When enabled, the PropertyManager tab  becomes active if you select a sketch entity.

**You can use PropertyManager to modify sketch entities in the following ways:**

- You can use the **Move/Copy** option on the PropertyManager window to translate or duplicate a sketch entity.
- Select the **For construction** check box to change the selected entities to construction geometry.
- If a spline is selected, select the **Proportional** check box to change the spline to a proportional spline.

The *Information point*  on the PropertyManager tab informs you about the status of the sketch entity, such as **Under Defined** or **Fully Defined**.
























---

**NOTE:** To fully define a sketch entity, you must still add relations or dimensions to the sketch entity. The PropertyManager does *not* constrain a sketch entity.

---

## Chapter 2 Sketching

You can edit the following values, as necessary, for each sketch entity:

Entity	Property
Point	X and Y coordinates of the point
Line	 X and Y coordinates of the start point
	 X and Y coordinates of the endpoint
	 Length
	 Angle
	 Delta X
	 Delta Y
Arc	 X and Y coordinates of the center point
	 X and Y coordinates of the start point
	 X and Y coordinates of the endpoint
	 Radius
	 Included angle
Circle	 X and Y coordinates of the center point
	 Radius
Ellipse	 X and Y coordinates of the center point
	 X and Y coordinates of the major axis point of the ellipse
	 X and Y coordinates of the minor axis point of the ellipse
	 Width of the minor axis
	 Width of the major axis
	 Included angle
Parabola	 X and Y coordinates of the start point
	 X and Y coordinates of the endpoint
	 X and Y coordinates of the focus
	 X and Y coordinates of the vertex
Spline	X and Y coordinates of each spline point. To edit a different spline point, change the number in the box.

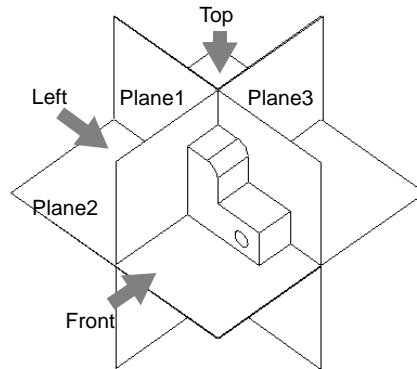
***To move or copy one or more sketch entities:***

- 1 Select one or more sketch entities.  
The PropertyManager tab appears.
- 2 Click **Move/Copy** to display the **Translate** and **Rotate** values on the PropertyManager tab.
- 3 Select the method to translate or rotate the selected sketch entities.
  - **Solve move.** All constraints are solved on the entities during the move.
  - **No solve move.** Move entities without solving dimensions or relations. See also **No Solve Move** on page 2-23.
  - **Copy.** Create a copy of the selected entities.
- 4 Edit the values to translate and rotate the selected sketch entities.
  - **Translate.** Type Delta X and Delta Y values to translate the selected entities a specified distance.
  - **Position selected point.** Type X and Y values to translate a selected point or vertex to a specified coordinate.
  - **Rotate.** Type an angle value to rotate the selected entities. Type X and Y values to change the center of rotation.
- 5 Click **Apply** to move or copy the selected entities.

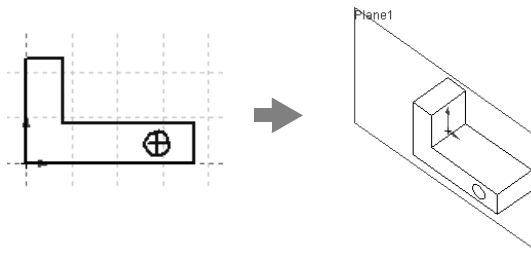
## Does it matter where I start sketching?

When you create a new part or assembly, the three default planes are aligned with specific views. The plane you select for your first sketch determines the orientation of your part.

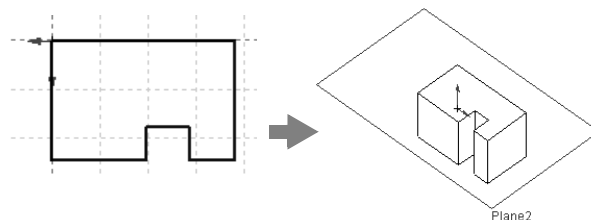
For example, if you choose **\*Front** in the **View Orientation** dialog box (or add a front view to a drawing), the view is normal to **Plane1**.



If you open a new part and start sketching *without* selecting a plane, the sketch is on **Plane1** by default and it is a *front* view.



If your first sketch is a top view, you should select **Plane2** in the FeatureManager design tree *before* you click the **Sketch** tool.



If your first sketch is a left or right view, select **Plane3**.

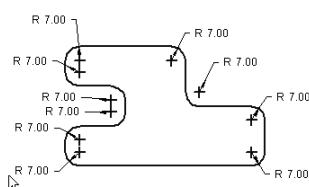
You do not have to use one of the default planes for your first sketch; you can create a new plane at any angle. The orientation of views is still determined by the default planes, however.

If you make a mistake or change your mind, you can reorient the part (to change “Front” to “Top” for example) with the **Update** button in the **View Orientation** dialog. For more information, see **Using Named Views** on page 1-29.

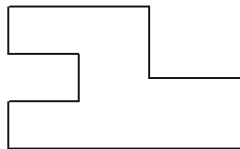
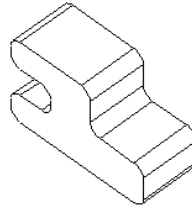
## How complex should sketches be?

In many cases, you can produce the same result by creating an extruded feature with a complex profile, or an extruded feature with a simpler profile and some additional features. (You often face this choice when planning the base feature for a part.)

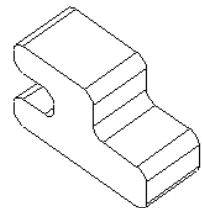
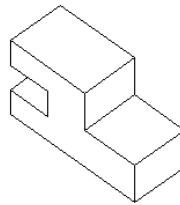
For example, if the edges of an extrusion need to be rounded, you can draw a complex sketch that contains sketch fillets (**A**), or draw a simple sketch and add the fillets as separate features later (**B**).



A. Complex sketch



B. Simple sketch



Add fillet features

Here are some things to consider:


- Complex sketches rebuild faster.** Sketch fillets can be recalculated much faster than fillet features, but complex sketches can be harder to create and edit.
- Simple sketches are more flexible and easier to manage.** Individual features can be reordered and suppressed, if necessary.







## 3D Sketching

---

Using SolidWorks 99, you can create 3D sketches. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a piping system. (For more information about piping, see Chapter 13, “Piping.”)

To begin a 3D sketch, click  on the Sketch toolbar, or click **Insert, 3D Sketch**. The sketch tools that are available in 3D sketching are **Line**, **Centerline**, **Fillet**, **Point**, and **Split Curve**.

3D sketching consists of lines and arcs in series. You use the **Line** tool  for sketched lines, and use the **Fillet** tool  to round the intersections of sketched lines. To create an arc, select the **Fillet** tool and click two intersecting line segments, or click their common point.

You can use the **Centerline** tool  and the **Point** tool  for construction geometry.

- ❑ **The coordinate system.** By default, you sketch relative to the default coordinate system in the model. To switch to one of the other two default planes, click the desired sketch tool, and press the **Tab** key. The origin of the current sketch plane is displayed.
  - To change the coordinate system for your 3D sketch, click the desired sketch tool, hold down the **Ctrl** key, and click a plane, a planar face, or a user-defined coordinate system.
  - If you select a plane or a planar face, the 3D sketch planes rotate so that the XY sketch plane is aligned with the selected item.
  - If you select a coordinate system, the 3D sketch planes rotate so that the XY sketch plane is parallel to the XY plane of the coordinate system.
- ❑ **Space Handle.** A graphical assistant is provided to help you maintain your orientation while you sketch on several planes. This assistant is called a *space handle*. The space handle displays when the first point of a line is defined on a selected plane. Using the space handle, you can select the axis along which you want to sketch.
- ❑ **Dimensioning.** You can sketch lines to the approximate length, and then dimension them exactly.
  - You can add a length dimension by selecting two points, a line, or two parallel lines.
  - You can add an angular dimension by selecting either three points or two lines.

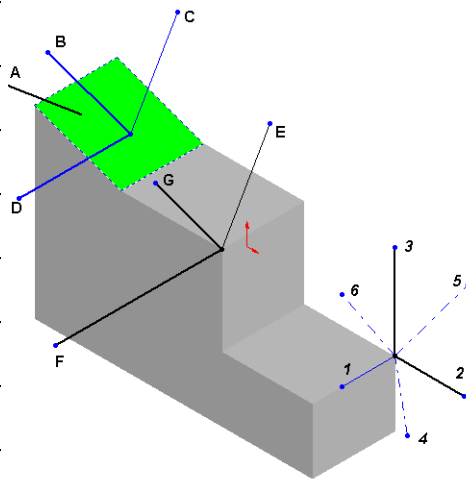
- **Geometric Relations.** Relations are automatically added for 3D sketch lines.
  - As you sketch a line, the line snaps to one of the major directions, X, Y, or Z, if applicable, and is constrained as **Horizontal**, **Vertical**, or **Along Z**, respectively. The relations are added with respect to the current coordinate system for the 3D sketch.
  - You are not restricted to drawing lines along one of the three major directions. You can sketch in the current sketch plane at an angle to one of the major directions, or you can sketch out-of-plane if the endpoint of the line snaps to existing model geometry.
  - You can add relations to points and lines in a 3D sketch. The table that follows shows some of the relations available with 3D sketching.

**Relations available in 3D sketching:**

<b>Relation</b>	<b>Selections</b>	<b>Result</b>
<b>Coincident</b>	point and plane	The point lies on the selected plane.
<b>Coincident</b>	line and plane	The line lies on the selected plane
<b>Horizontal</b>	line and plane	The line lies parallel to the horizontal direction of the selected plane.
<b>Vertical</b>	line and plane	The line lies parallel to the vertical direction of the selected plane.
<b>Along Z</b>	line and plane	The line lies parallel to the normal direction of the selected plane.
<b>Perpendicular</b>	line and plane	Same as <b>Along Z</b> .
<b>Parallel</b>	line and plane	The line lies parallel to the selected plane.
<b>Parallel YZ</b>	line and plane	The line lies parallel to the YZ plane with respect to the selected plane.
<b>Parallel ZX</b>	line and plane	The line lies parallel to the ZX plane with respect to the selected plane.

**Graphic view of 3D sketching relations:**

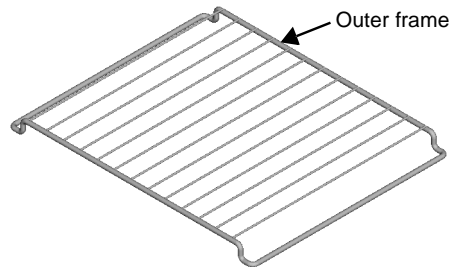
<b>A</b>	Selected plane for lines <b>B - F</b>
<b>B</b>	Sketched along plane X direction horizontal constraint, line-plane
<b>C</b>	Sketched along normal to plane along Z constraint, line-plane
<b>D</b>	Sketched along plane Y direction vertical constraint, line-plane
<b>E</b>	Sketched parallel to normal to plane along Z constraint, line-plane
<b>F</b>	Sketched parallel to plane Y direction vertical constraint, line-plane
<b>G</b>	Sketched parallel to plane X direction horizontal constraint, line-plane
<b>1</b>	Sketched along global Z along Z constraint
<b>2</b>	Sketched along global X horizontal constraint
<b>3</b>	Sketched along global Y vertical constraint
<b>4</b>	Sketched in global ZX plane no constraint
<b>5</b>	Sketched in global XY plane no constraint
<b>6</b>	Sketched in global YZ plane no constraint



- ❑ **Line snap.** During line creation of a line, you can snap the line to geometry that already exists in the part, such as model surfaces or vertices, and sketch points.
  - Snap is not enabled if you are sketching in one of the major coordinate directions.
  - If you are sketching on a plane, and the system infers a snap to a non-planar point, a temporary 3D graphics box is displayed to indicate an off-planar snap.
- ❑ **Virtual sharps.** When you create an arc with the **Fillet** tool, you can display the original common point of the lines as a *virtual sharp*. To display a point at the virtual sharp, click **Tools, Options, General**, and select **Display virtual sharps** in the **Sketching** section of the **General** dialog box.
  - You can add dimensions and relations to virtual sharps in a 3D sketch.
  - If you delete an arc, the lines extend to meet at the virtual sharp.
  - If you delete a line used to create a fillet, the arc it is joined with is also deleted. The other line extends to the virtual sharp (the original common point).

### 3D sketching example:

Shown here is an oven rack assembly. The outer frame of the assembly is built by sweeping a circle along a 3D sketch, then mirroring the sweep feature.






#### To create the 3D sketch of the outer frame:


- 1 Open a new part, and change the view orientation to **Isometric**.




---


**TIP:** It is easier to create a 3D sketch in an isometric orientation because the X, Y, and Z directions are visible.

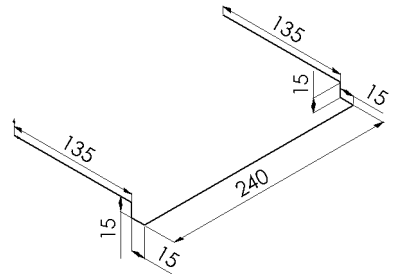
---

- 2 Click **3D Sketch**  on the Sketch toolbar, or click **Insert, 3D Sketch** to open a new sketch.
- 3 Click **Line**  and sketch a 135mm line on the XY plane  from the origin along the X axis.

The pointer changes to  while sketching along the X axis.

- 4 Continue sketching the other lines from the endpoint of the previous line. Each time you begin a new line, the origin for the current coordinate system is displayed at the beginning of the new line to help orient you as you sketch.
  - a) Sketch down the Y axis  for 15mm.
  - b) Sketch along the X axis for 15mm.
  - c) Press **Tab** to change the sketch plane to the YZ plane .
  - d) Sketch along the Z axis  for 240mm.
  - e) Press **Tab** twice to change the sketch plane back to the XY plane.
  - f) Sketch back along the X axis for 15mm.
  - g) Sketch up the Y axis for 15mm.
  - h) Sketch back along the X axis for 135mm.

- 5 Dimension each of the lines as shown.
- 6 Click **Fillet**  on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Fillet** and fillet each intersection with a 5mm fillet.
- 7 Close the sketch.



To complete the outer frame:

- Sweep a 5mm diameter circle along the 3D sketch path.
- Mirror the feature to create the other half of the frame.

## Reference Geometry

---

Reference geometry defines the shape or form of a surface or a solid. Reference geometry includes planes, axes, coordinate systems, and 3D curves. In Chapter 4, “Creating Features,” you can see how reference geometry is used in the creation of several kinds of features: construction planes are used in lofts and sweeps, split lines are used in some drafts and chamfers, an axis is used in a circular pattern, and so on.


This chapter describes how to create and use reference geometry in the SolidWorks 99 software, and describes the following:

- Creating and using construction planes
- Creating an axis
- Creating local coordinate systems
- Converting sketch lines to construction geometry
- Creating 3D curves
- Displaying reference geometry

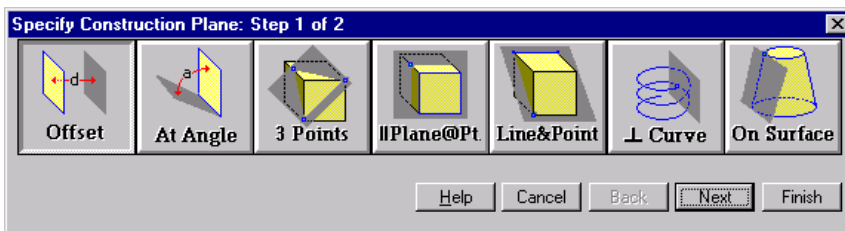
## Creating a Construction Plane

By default, a new sketch opens on **Plane1**. You can also select **Plane2** and **Plane3** for sketching, as discussed on page 2-26. Additionally, you can create new planes to sketch on.

### To create a plane:

- 1 Click **View, Planes**.
- 2 Click  on the Reference Geometry toolbar, or **Insert, Reference Geometry, Plane**.
- 3 Select the plane type from the **Specify Construction Plane** dialog box, and click **Next**.

The following plane types are available:



**Offset.** Creates a plane parallel to a plane or face, offset by a specified distance.

- 1 Select a plane or a planar face.
- 2 Specify the offset **Distance**.
- 3 Click **Reverse Direction**, if necessary.
- 4 Click **Finish**.

**At Angle.** Creates a plane through an edge, axis, or sketch geometry at an angle to a face or plane.

- 1 Select a plane or a planar face.
- 2 Select an edge, axis, or sketch line.
- 3 Specify the **Angle** between the new and existing plane.
- 4 Click **Reverse Direction**, if necessary.
- 5 Click **Finish**.

**Three Point Plane.** Creates a plane through three points.

- 1 Select three vertices, points, or midpoints.
- 2 Click **Finish**.

---

**NOTE:** To select a *midpoint*, right-click an edge then click **Select Midpoint**. A cross appears in the middle of the edge.

---

**Parallel Plane at Point.** Creates a plane through a point parallel to a plane or face.

- 1 Select a plane or planar face and a vertex, point, or midpoint.
- 2 Click **Finish**.

**Line and Point.** Creates a plane through a line, axis, or sketch line and a point.

- 1 Select an edge, axis or sketch line and a vertex, point, or midpoint.
- 2 Click **Finish**.

**Perpendicular to Curve at Point.** Creates a plane through a point and perpendicular to an edge, axis, or sketch curve.

- 1 Select an edge, axis, or sketch curve and a vertex or point.
- 2 Click **Finish**.

**On Surface.** Creates a plane on a non-planar surface.

- 1 Select a surface.
- 2 Select a plane that intersects that surface.
- 3 If no solution is displayed, you need to select an edge on that surface.
- 4 Click **Other Solutions**, if necessary.
- 5 Click **Finish**.


**Plane Tangent to Surface at Point.** Creates a plane that is tangent to a non-planar surface at one of the following points:

- a point projected onto the surface along the normal direction of the sketch plane from a sketch point, or the endpoint of a sketch entity

– or –

- on the surface closest to a selected sketch point, a vertex, the end of a curve, or an endpoint of a sketch entity.

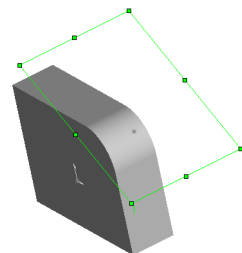
**Example of a plane tangent to a surface at a sketch point:**

- 1 Open a sketch on a face of a part and place a sketch point to project onto the non-planar surface. Close the sketch.
- 2 Click **Plane**  on the Reference Geometry toolbar, or **Insert, Reference Geometry, Plane**.
- 3 Click **On Surface** in the **Specify Construction Plane** dialog box, and click **Next**.
- 4 Click the non-planar surface and the sketch point in the graphics area.
- 5 Select the **Project sketch point along sketch plane normal** check box.

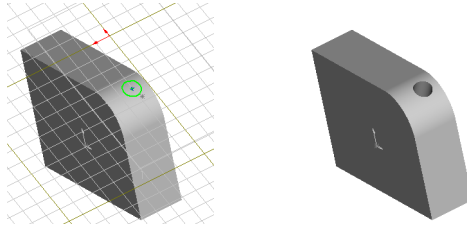
The preview of the plane appears. The plane is tangent to the non-planar surface at the point where the sketch point is projected along the normal direction of the point's sketch plane.

– or –

If you do *not* select **Project sketch point along sketch plane normal**, the minimum distance solution is used.



- 6 Select the **Reverse direction** check box, if desired, to create the plane on the opposite side of the sketch plane. (May not be an appropriate choice in all cases.)
- 7 Click **Finish**.  
Now you can open a sketch on the plane and use it to make a feature on the non-planar surface.



## Moving, Resizing, and Copying Planes

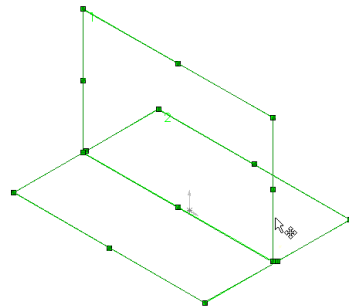
You can move, resize, and copy planes using the plane handles and edges.

### *To display the plane's handles:*

- Click the plane's name in the FeatureManager design tree or in the graphics area,  
– or –
- Click the edge of the plane.

### *Using the plane's handles and edges, you can do the following:*

- Resize the plane by dragging a corner or edge handle.
- Move the plane by dragging the edge of the plane.
- Copy the plane by selecting a plane in the graphics area. Then hold down the **Ctrl** key and, using the *edge*, drag to a new location.



### *To modify the offset distance, angle, or distance between planes:*

- 1 Double-click the plane to display the offset distance or angle.
  - 2 Double-click the dimension or angle and type a new value in the **Modify** dialog box.
- or –
- 1 Right-click the name of the plane in the FeatureManager design tree.
  - 2 Select **Edit Definition**.
  - 3 Type new values to define the plane, and click **Finish**.



## Changing Plane Names

### ***To change the names for construction planes in the current document:***

- 1 Click-pause-click the plane's name in the FeatureManager design tree.
- 2 Type a new name.

As you create additional construction planes, it is a good idea to change their names to indicate their purpose.

### ***To change the default names used for planes:***

You can create new default plane names to replace the names, Plane1, Plane2, and Plane3. The new names will be used on all new parts and assemblies. For example, you may want to name the planes Front, Top, and Right, instead of Plane1, Plane2, and Plane3.

- 1 Click **Tools, Options, Reference Geometry**.
- 2 In the **Plane default names** section, type new names in the boxes that correspond to the original plane names.
- 3 Click **OK**.

## Hide or Show Planes

You can turn the display of planes on or off.

### ***To turn plane display on or off:***

Click **View, Planes**. A check mark next to the menu item means planes are visible (except for planes you have hidden individually).

### ***To hide or show individual planes:***

- 1 Right-click the plane or its name in the FeatureManager design tree.
- 2 Click **Hide** or **Show**.

---

**NOTE:** Individual planes always are highlighted when you select them, even when hidden.

---

## Changing the Plane of a Sketch

For information about changing the sketch plane of a model, see **Edit Sketch Planes** on page 2-32.

## Creating an Axis

---

You can use an axis in creating sketch geometry.

Every cylindrical and conical face has an axis. Temporary axes are those created implicitly by cones and cylinders in the model. You can use a temporary axis to insert an axis in a model.

You can set the default to either hide or show all temporary axes.

### ***To display temporary axes:***

Click **View, Temporary Axes**.


### ***To turn the display of axes on or off:***

Click **View, Axes**. A check mark next to the menu item means axes are visible (except for axes you have hidden individually).

### ***To hide or show an individual axis:***

- 1 Right-click the axis or its name in the FeatureManager design tree.
- 2 Click **Hide** or **Show**.

### ***To create an axis in a part or assembly:***

- 1 Click **Axis**  on the Reference Geometry toolbar, or **Insert, Reference Geometry, Axis**.
- 2 Select from the **Defined by** box:
  - **One Line/Edge/Axis**. Select a sketched line, a model edge, or select **View, Temporary Axes**, and then select the axis that is displayed.
  - **Two Planes**. Select two intersecting construction planes.
  - **Two Points/Vertices**. Select two sketched points, two vertices, or a point and a vertex.
  - **Cylindrical/Conical Surface**. Select a conical or cylindrical face.
  - **Point and Surface**. Select a plane or planar surface and a sketched point or a vertex.
- 3 Verify that the items listed in the **Selected items** box correspond to your selections.
- 4 Click **OK**.

---

**NOTE:** To see the new axis, click **View, Axes** to turn on axes display.


---

## Creating a Coordinate System

---

You can define a coordinate system for a part or assembly. Use this coordinate system with the **Measure** and **Mass Properties** tools, and for exporting SolidWorks documents to IGES, STL, ACIS, STEP, VDA and Parasolid. See Chapter 14, “Importing and Exporting Files.”

### *To create a coordinate system:*

- 1 Click **Coordinate System**  on the Reference Geometry toolbar, or **Insert, Reference Geometry, Coordinate System**.
- 2 Select a vertex, sketched point, midpoint of an edge, or the default point of origin on a part (or component, in an assembly). The name of the entity appears in the **Origin** box.
- 3 Click in the box for the **X Axis**, **Y Axis**, or **Z Axis**, and then click an edge, face, or vertex on the part, or a sketch segment to indicate the direction of the selected axis.
- 4 Click another axis and select another edge, face, or vertex to define this axis.
- 5 If you need to change your selections, right-click in the graphics area and select **Clear Selections**.
- 6 If you need to reverse the direction of an axis, click the **Flip** check box under its definition.
- 7 When the coordinate system is defined, click **OK**.

---

**NOTE:** When you create a coordinate system, it is a good idea to give it a meaningful name to explain its purpose. Click-pause-click the coordinate system's name in the FeatureManager design tree and type a new name.


---

### *To edit definition of the coordinate system:*

Right-click the name of the coordinate system in the FeatureManager design tree and select **Edit Definition**.

### *To move a coordinate system to a new location:*

You may need to locate a coordinate system where there are no edges, vertices, or points available to control the definition of the angle or direction of an axis. In this case, you can define a coordinate system some place on the part that does provide the entities you need. Then you can move the new origin to the desired location.

- 1 Click **Coordinate System**  or **Insert, Reference Geometry, Coordinate System**.
- 2 Define the coordinate system at a location on the part that provides the entities you need to control the angle and direction of each axis.
- 3 Click the **Origin** box and click the location to which you want to translate the origin. The new origin moves to the location that you selected.
- 4 Click **OK**.

## Converting Sketch Lines to Construction Geometry

---

You can convert sketched lines, arcs, and splines into construction geometry to use in creating model geometry.

***To convert a single sketch entity to construction geometry:***

- 1 Right-click the sketch entity and select **Properties**.
- 2 On the PropertyManager tab, click the **For construction** check box.

---

**NOTE:** If you have enabled the directing editing of a sketch, the properties of selected sketch entities are always available on the PropertyManager tab. In this case, to convert a sketch entity to construction geometry, select the sketch entity and click the **For construction** check box on the PropertyManager tab.

To enable direct editing, click **Tools, Options, General**, and select the **Enable direct edit of sketch** check box.

---

***To convert several sketch entities to construction geometry at the same time:***

- 1 Hold down the **Ctrl** key and right-click the sketched entities.
- 2 Select **Properties** from the shortcut menu.
- 3 In the **Common Properties** dialog, click the **Construction Geometries** check box.
- 4 Click **OK**.

***To convert sketch entities in a drawing to construction geometry:***

- Select the sketch entities and click **Tools, Sketch Entity, Construction Geometry**.
- Right-click one sketch entity and select **Properties** in the shortcut menu. On the PropertyManager tab, click the **For construction** check box.

For more information about using sketch entities in a drawing, see **2D Sketching** on page 9-39.

## Curve

---

A *curve* is a type of geometry. You can create several types of 3D curves by using these methods:

- Projected Curve** - from a sketch projected to a model face or surface, or from sketched lines on intersecting planes
- Composite Curve** - a combination of curves, sketch geometry, and model edges into a single curve
- Helix** and **Spiral** - by specifying a circular sketch, pitch, number of revolutions, and height
- Split Line** - from a sketch projected to planar or curved (silhouette) faces
- Curve Through Reference Points** - from user-defined points or existing vertices
- Curve Through Free Points** - from a list of X,Y, Z coordinates for points

You can then use the curves to create solid model features. For example, you can use a curve as the path or guide curve for a sweep feature, as the guide curve for a loft feature, as a parting line for a draft feature, and so on.

### Projected Curve

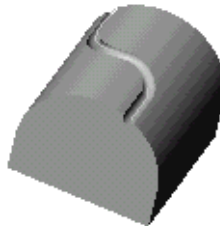
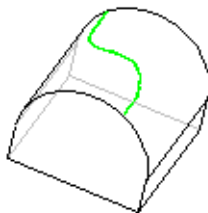
You can project a sketched curve onto a model face to create a 3D curve.

#### *To project a curve onto a face:*

- 1 Create a sketch containing a single open or closed curve (made up of lines, arcs, or splines) on a plane or model face.
- 2 Close the sketch.
- 3 Select the sketch, hold down the **Ctrl** key and select the face where you want to project the curve.
- 4 Click **Insert, Curve, Projected**.



The curve appears in the FeatureManager design tree and on the selected face.



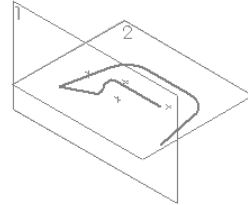
Projected curve used as a sweep path

You also can create a 3D curve that represents the intersection of two extruded surfaces generated by creating sketches on two intersecting planes.

### **To create a projected curve using sketches on intersecting planes:**

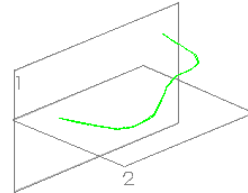
- 1 Create a sketch on each of two intersecting planes, closing each sketch when you are done.

Align the sketch profiles such that when they are projected normal to their sketch plane; the implied surfaces will intersect, creating the desired result.

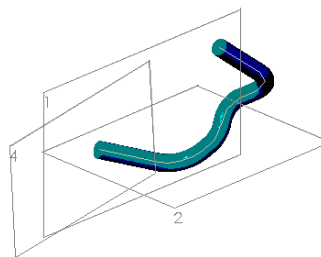
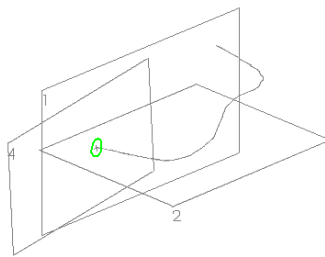


- 2 Click **Insert, Curve, Projected**.

The system creates the projected curve.



**Example:** The following is an example of using a 3D projected curve to create a swept feature. Note the addition of a profile sketch for the sweep on plane 4.



For more information about creating sweep features, see **Sweep** on page 4-10.

## Replacing a Projected Curve

You can replace one or more of the sketches that make up a projected curve of two sketches projected onto one another.

### **To replace a sketch in a projected curve:**

- 1 Right-click the **Curve** feature in the FeatureManager design tree, or right-click the curve in the graphics area, and select **Edit Definition**.

The **Project sketch onto sketch** dialog box appears.

- 2 Choose one of the following methods to clear the unwanted sketch from the **Sketch selected** box:


- Click the sketch in the FeatureManager design tree.
- Click the sketch in the graphics area.
- Select the sketch in the **Sketch selected** box, and press **Delete**.

- 3 Select the replacement sketch in either the FeatureManager design tree or in the graphics area, then click **OK**.

## Composite Curve

You can create composite curves by combining curves, sketch geometry, and model edges into a single curve. Use this curve as a guide curve when creating a loft or a sweep.

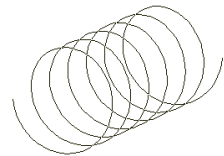
### *To create a composite curve:*

- 1 Click **Composite Curve**  on the Reference Geometry toolbar, or **Insert, Curve, Composite**.
- 2 Click the items (sketch entities, edges, and so on) that you want to join. The list of the selected items displays in the **Items to join** box.
- 3 Click **OK** to create the composite curve.

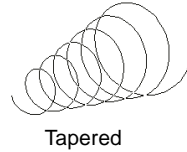
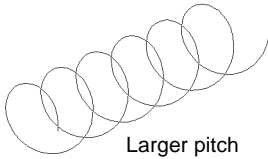
## Helix

### *To create a helix:*

- 1 Open a sketch and sketch a circle. The diameter of this circle controls the diameter of the helix.
- 2 Close the sketch, and select the circle.
- 3 Click **Insert, Curve, Helix/Spiral**.
- 4 Select one of the following definitions from the **Defined by** scroll box. (To define a helix, you specify two values, and the third is calculated automatically.)
  - **Pitch and Revolutions**
  - **Height and Revolutions**
  - **Height and Pitch**
- 5 Depending on the definition, specify these values:
  - **Height** - the parallel distance between the endpoints of the helix
  - **Pitch** - the parallel distance required for one full revolution
  - **Revolution** - the number of turns in the helix
- 6 If you want a tapered helix, click **Taper Helix**, and specify a taper **Angle**. The **Taper outward** option determines whether the helix tapers inward or outward.



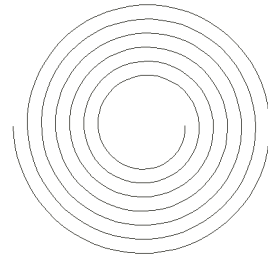
- 7 If necessary, click **Reverse direction**, modify the **Starting angle** (on the plane of the sketched circle), and choose the direction of the turns (**Clockwise** or **Counterclockwise**).



- 8 Click **OK**.

**To create a spiral:**

- 1 Open a sketch and sketch a circle. The radius of the sketched circle is the radius of the spiral at its start point.
- 2 Close the sketch.
- 3 Select the circle.
- 4 Click **Insert, Curve, Helix/Spiral**.
- 5 In the **Defined by** scroll box select **Spiral**.
- 6 Type a value for the **Pitch** - this determines the rate of change of radius for each revolution.



---

**NOTE:** The **Pitch** value must be greater than 0.0 and not greater than 200000.

---

- 7 Specify the number of **Revolutions** - the number of turns in the spiral. (This does not have to be a whole number.)
- 8 Select the **Reverse direction** check box to create an inward spiral.
- 9 Specify a value in the **Starting angle** box to indicate where to start the spiral.
- 10 To choose the direction of the turns, click **Clockwise** or **Counterclockwise**.
- 11 Click **OK**.

## Split Line

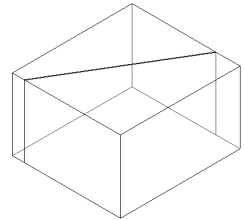
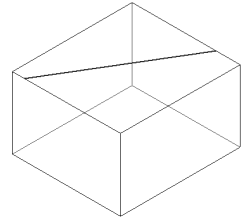
**Split Line** projects a sketched curve onto selected model faces. It divides a selected face into multiple separate faces so that each can be selected. **Split Line** can be used for a variety of purposes:

- When creating a **Draft** feature with the **Parting Line** option. (For more information, see **Parting Line** on page 4-30 or **Creating Molds** on page 7-18.)
- When creating a face blend fillet using a hold line. (See **Face Blend with Hold Line** on page 4-26.)
- When creating a **Radiate Surface**. (See **Radiate Surface** on page 4-53.)



**To project a split line onto a planar face:**

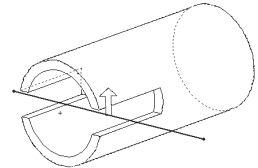
- 1 Open a sketch on the face of the part and sketch a line to project as a split line.
- 2 Exit the sketch.
- 3 Click **Insert, Curve, Split Line**.
- 4 In the **Split Lines** dialog box, select **Projection** and click **Next**.
- 5 In the **Project Split Line** dialog, click the **Sketch to project** box and select the sketch in the FeatureManager design tree.
- 6 Click the **Faces to split** box, then select all the faces around the perimeter of the part that you want the split line to pass through.
- 7 Click **Finish**.



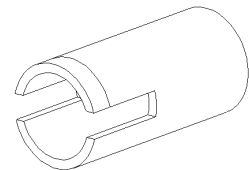
In special cases where there is an interruption in the selected face, you can project the curve in a single direction, and specify the direction of pull.

In this example, the **Face to split** is cylindrical, and the sketch plane of the **Sketch to project** lies in the area of a cut.

- Click **Single direction** to project the curve in one direction only.
- Click **Reverse direction** if the preview indicates that the curve projects the wrong way.

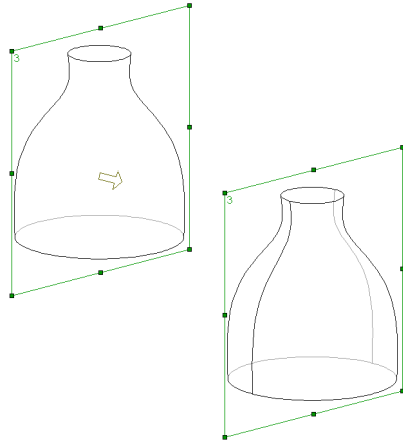


If you do not click **Single direction**, the curve projects all the way around the cylinder.



**To create a silhouette split line on a part with curved or tangent faces:**


- 1 With a part open, click **Insert, Curve, Split Line**.
- 2 In the **Split Lines** dialog box, select **Silhouette** and click **Next**.
- 3 Click the **Direction of pull** box.
- 4 In the FeatureManager design tree, click a plane, edge, face, or axis that extends lengthwise through the model.
- 5 Click the **Faces to split** box, then select the curved or tangent faces to split.
- 6 Click **Finish**.



### Curve Through Reference Points

Creates a 3D spline through points located on one or more planes. You can use sketch points or model vertices.

**To create a curve through reference points:**

- 1 Click **3D Curve** , or **Insert, Curve, Curve Through Reference Points**.  
The **Curve** dialog box appears.
- 2 Select the points in the order in which you want to create the curve.  
As you select, the number of sketch items and vertices is updated in the **Spline Points** box, and a preview of the curve is displayed.
- 3 If you want to close the curve, click the **Closed curve** box.
- 4 Click **OK**.

## Curve Through Free Points

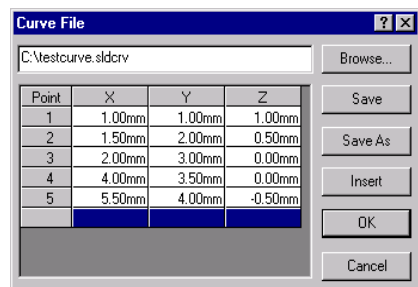
You can create a 3D curve from a set of points. There are two ways to generate a point list for this purpose:

- ❑ You can create a curve one point at a time, observing the effect on the resulting curve as you go along.
- ❑ You can create a list of points using a text editor or other tool, such as a spreadsheet program.

### To create a curve by specifying coordinate values:

- 1 Click **Insert, Curve, Curve Through Free Points**.
- 2 In the **Curve File** dialog box, double-click the first cell in the first row.
- 3 Type the coordinates for **X**, **Y**, and **Z**. Press the **Tab** key to move from cell to cell, and from row to row.

Notice that the value in the first cell in the **Point** column is set to 1. The numbers in the **Point** column define the order in which the points will be connected.



- 4 Continue adding as many points as needed (one row is one point). The larger the number of points, the greater your control of the resulting curve.

Notice the preview of the curve in the graphics area.

- 5 If necessary, you can insert a new point between existing points. Click the point number to select the row *below* where you want the new point to occur, then click **Insert**. A new row is added *above* the row you clicked. All the points below the selected row move down.
- 6 Double-click in a previously filled cell to go back and change its value, if necessary.
- 7 You can save the curve file for re-use. Click **Save** or **Save As**, navigate to the desired location, and specify the filename. If you do not specify an extension, the software adds the extension **.sldcrv**.
- 8 Click **OK** to complete the curve and close the dialog box.

### **To edit a curve file:**

Right-click the curve name in the FeatureManager design tree, or right-click the curve in the graphics area. Select **Edit Definition** from the shortcut menu.

### **To create a curve from an existing file containing coordinate values:**

- 1 Create a point file using a text editor, worksheet, or other utility. The format of the file should be a three-column, tab- or space-delimited list of X, Y, and Z coordinates.

If you use a worksheet to create the file, save it as *text*, with either tabs or spaces as the field delimiter.

- 2 Click **Insert, Curve, Curve Through Free Points**.
- 3 Use the **Browse** button to locate the file, then click **Open**.

The coordinates from the file are displayed in the **Curve File** dialog box. Notice the numbers in the **Points** column, indicating the order in which the points will be connected.

- 4 Examine the preview of the curve, and edit the coordinates if necessary, until you are satisfied with the result. (Double-click in any cell, then edit the value.)
- 5 Click **OK**.

## Displaying Reference Geometry

---

Reference geometry is a necessary tool, but when it is visible, it may make the image of the part appear too complex and hard to understand. You can set system-wide visibility options to filter out various types of reference geometry. Click **Tools, Options, Reference Geometry** and select the kinds of reference geometry that you want to see, by default. See *Appendix A, Reference Geometry Options* on page A-20 for more information.

To set the visibility of reference geometry in the current document, use the options under the **View** menu.

By default, reference geometry is hidden in drawing views. For information about importing reference geometry to drawings, see **Reference Geometry in a Drawing** on page 10-17.

## Creating Features

---

Features are the individual shapes that, when combined, make up the part. You can also add some types of features to assemblies. Some features originate as sketches; other features, such as shells or fillets, are created when you select the appropriate menu command and define the dimensions or characteristics that you want.

Surfaces can be used to create or modify solid features. Surfaces can be imported from other applications and thickened or used in other ways to create solid geometry. Surfaces can be created in ways similar to those used to create solid features. You can extrude, revolve, sweep, or loft a surface. You can also knit, or offset from a model face or surface. A surface may have either an open or closed profile. See **Surfaces** on page 4-50.

This chapter describes the following:

- Features toolbar
- Base, Boss, and Cut
- Extrude, Revolve, Sweep, and Loft
- Fillet/Round, Chamfer, and Draft
- Hole - Simple and Hole Wizard
- Scale
- Shell
- Rib
- Dome
- Shape
- Pattern and Mirror
- Surfaces

## The Features Toolbar

---

For many of the features described in this chapter, there is an icon on the Features toolbar. For others, an icon may be available, but not displayed by default. You can customize the toolbars to include icons for some of the functions you use most.

Click **Tools, Customize**, and click the **Commands** tab. Choose a category, and examine the available buttons. Drag the buttons you want into any toolbar, delete buttons you do not want from any toolbar, or rearrange buttons to your liking.

## Base/Boss

---

The first feature of every part created is the *base* feature. (There is only one *base* feature.)

A *boss* is a feature that adds material to a part.

A *base* or *boss* may be created by **Extrude**, **Sweep**, **Revolve**, or **Loft** (from one or more sketches), or **Thicken** (from a surface).

For information about surfaces, see **Surfaces** on page 4-50.

## Cut

---

A *cut* is a feature that removes material from a part or an assembly.

A cut may be created by **Extrude**, **Sweep**, **Revolve**, or **Loft** (from one or more sketches), or **Thicken** or cut **With Surface** (from a surface).

## Extrude

---

*Extrude* extends a sketched profile in one or two directions as either a *thin feature* or a *solid feature*.

An extrude operation can either add material to a part (in a base or boss) or remove material from a part (in a cut or hole).



Solid Feature

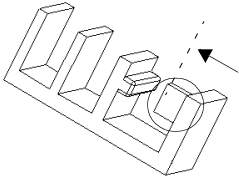
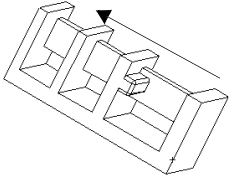
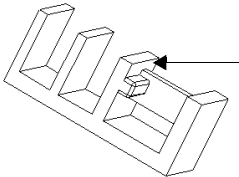
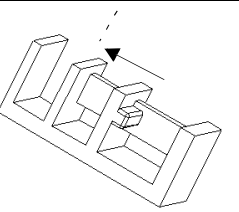
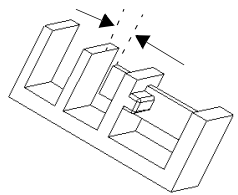


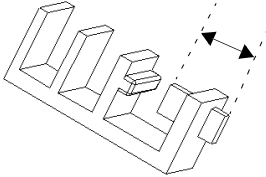
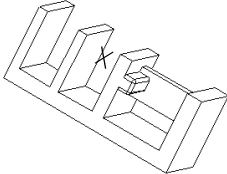
Thin Feature

The **End Condition** tab of the **Extrude Feature** dialog lets you define the characteristics of extruded features created with these commands on the **Insert** menu:

- **Base, Extrude**
- **Boss, Extrude**
- **Cut, Extrude**
- **Features, Hole, Simple**

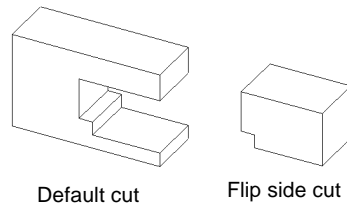
The following table describes the **End Condition** tab of the **Extrude Feature** and the **Extrude Cut Feature** dialog boxes and the various options for extruded bosses and cuts.

Type	Example	Description
<b>Blind</b>		Extends the feature from the sketch plane for a specified distance ( <b>Depth</b> ).
<b>Through All</b>		Extends the feature from the sketch plane through all existing geometry.
<b>Up To Next</b>		Extends the feature from the sketch plane to the next face, or set of faces, that intercepts the entire profile. The <i>next</i> face must be on the same part.
<b>Up To Surface</b>		Extends the feature from the sketch plane to the selected face or surface.
<b>Offset From Surface</b>		Extends the feature from the sketch plane to a specified distance from the selected face or surface.

Type	Example	Description
Mid Plane		<p>Extends the feature from the sketch plane equally in both directions. (<b>Depth</b> specifies the <i>total</i> depth, not the depth in each direction.)</p>
Up To Vertex		<p>Extends the feature from the sketch plane to a plane that is parallel to the sketch plane and passing through the specified vertex.</p>

### Specifying End Conditions

- Type** determines how far the feature extends. See the preceding table for examples.
  - If you choose **Blind** or **Mid Plane**, you have to specify the **Depth**.
  - If you choose **Offset From Surface**, you have to specify the **Offset**. Click **Reverse Offset**, if appropriate.
- Depth** specifies the depth of the extrusion.
- Reverse Direction** lets you extend the feature in the opposite direction from that shown in the preview in the graphics area.
- Link to Thickness** is used primarily for bosses on sheet metal parts. Selecting this option automatically links the depth of an extruded boss to the thickness of the base feature.
- Flip Side to Cut** appears only when you are extruding a cut. By default, material is removed from the *inside* of the profile. Selecting **Flip Side to Cut** removes all material from the outside of the profile.



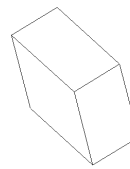

---

**NOTE:** If you are using a closed profile for the cut and you select the **Flip Side to Cut** check box, you should select **Through All** as the type of end condition.

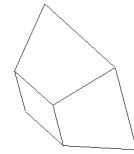
---



- ❑ **Draft While Extruding** lets you add draft to a feature while you extrude it. If you select this option, you must set the draft **Angle**, and you can select **Draft Outward**, if needed.
- ❑ **Selected Items.** If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is indicated in the **Selected Items** box.



No draft



25° draft angle – inward

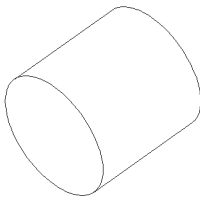
- ❑ One direction or **Both Directions**:
  - The feature is extruded in one direction from the sketch plane. This is the default (**Both Directions** *not selected*).
  - **Both Directions** extrudes the feature in both directions from the sketch plane. Specify all the settings for the first direction (**Direction 1**), then select **Direction 2** from the **Settings for** box, and specify the settings for the second direction.

---

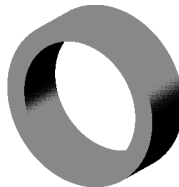
**NOTE:** Observe the preview to verify the direction and depth of the feature.

---

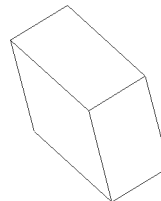
- ❑ **Extrude As:**
  - **Solid Feature** adds (or removes) solid volumes to the model.
  - **Thin Feature** adds (or removes) thin-walled volumes to the model. A **Thin Feature** base can also be used as a basis for a sheet metal part. For more information, see Chapter 12, “Sheet Metal.”



A circle extruded as a solid feature



A circle extruded as a thin feature



A rectangle extruded as a solid feature



A rectangle extruded as a thin feature, with draft

## Thin Features

The following menu items are for thin features only, and appear when you choose the **Thin Feature** tab.

---

**NOTE:** Whenever you sketch an open profile, the **Thin Feature** tab appears in the **Extrude Feature** dialog box. When you sketch a closed profile, you need to select **Thin Feature** in the **Extrude As** box to display the **Thin Feature** tab.

---

- Type** specifies whether to extrude the thin feature in **One-Direction**, **Mid-Plane**, or **Two-Direction**.
  - **One-Direction** extrudes the sketch in one direction using the specified wall thickness.
  - **Mid-Plane** extrudes the sketch in both directions, dividing the specified wall thickness equally on both sides of the sketch geometry.
  - **Two-Direction** extrudes the sketch in both directions, using a different wall thickness on each side of the sketch geometry (as specified for **Direction 1 Thickness** and **Direction 2 Thickness**).
- Wall Thickness** specifies the thickness of the thin feature wall.
- Reverse**. The default is to add the wall thickness to the outside of the sketched profile. Clicking **Reverse** adds the wall thickness to the inside of the sketched profile.

For thin feature base extrusions only, you can specify the following additional options:


- If you create a *closed profile* sketch, you can use the **Cap Ends** option. This option covers (caps) the ends of the feature, creating a hollow part. If you select this option, you must also specify the **Cap Thickness**.
- If you create an *open profile* sketch, you can use the **Auto Fillet** option. This option creates a round at each edge where lines meet at an angle. If you select this option, you must also specify the **Fillet Radius** (the inside radius of the round).

## Extruded Base/Boss



**To create an extruded base, boss, or surface:**

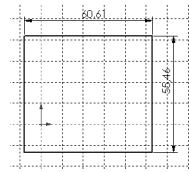
- 1 With a sketch active, click one of the following:

- **Extruded Base/Boss**  on the Features toolbar, or **Insert, Boss, Extrude** or **Insert, Base, Extrude**
- **Insert, Surface, Extrude**

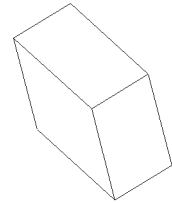
The **Extrude Feature** dialog box and a preview of an extrusion appear. In most instances, the preview updates automatically whenever you change any of the following options.

- 2 Select the **Type** of extrusion. (See the table on page 4-3 for more information.)
- 3 Set the desired **Depth** of the extrusion.  
For bosses on sheet metal parts, click **Link to Thickness** on the **Thin Feature** tab to link the depth of the boss to the thickness of the base feature.
- 4 To add a draft angle, click **Draft While Extruding**. Enter a value for **Angle**, and click **Draft Outward**, if desired.
- 5 If the **Type** you specified relies on the selection of a surface or vertex, click that item in the graphics area. The selection is indicated in the **Selected Items** box.
- 6 Examine the preview and click **Reverse Direction** to extrude in the opposite direction, if necessary.
- 7 To extrude the feature in two directions from the sketch plane, click **Both Directions**, select **Direction 2** in the **Settings for** box, and enter the settings for **Direction 2**. If you are extruding a surface, skip ahead to Step 10.
- 8 If creating a base or boss feature, select **Solid Feature** or **Thin Feature** in the **Extrude as** box. If you are extruding a **Solid Feature**, skip ahead to Step 10. If you are extruding a **Thin Feature**, proceed to Step 9.
- 9 Select the **Thin Feature** tab.
  - a) Select a wall thickness **Type**. (See the descriptions on page 4-6.)
  - b) Specify the **Wall Thickness**, and if you want to add the wall thickness to the opposite side of the sketched profile, click **Reverse**.
  - c) To add fillets to the corners of an open profile thin base feature, click **Auto Fillet**, and specify the desired **Fillet Radius**.
  - d) To cap the ends of a closed profile base feature, click **Cap Ends**, and specify the desired **Cap Thickness**.
- 10 When you are satisfied with the preview, click **OK**.

Sketched rectangle




Sketched rectangle extruded to make a box



## Extruded Cut

### To create an extruded cut:

- 1 With a sketch active, click **Extruded Cut**  on the Features toolbar, or click **Insert, Cut, Extrude**.

The **Extrude Cut Feature** dialog box appears.

- 2 Select the desired **Type**. (See the table on page 4-3 for more information.)

- 3 Specify the **Depth**.

- 4 Examine the preview and click **Reverse Direction**, if necessary.

- 5 Select **Flip Side To Cut**, if desired.

- 6 To add a draft angle, click **Draft While Extruding**. Enter a value for **Angle**, and click **Draft Outward**, if desired.

- 7 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is displayed under **Selected Items**.

- 8 To extrude the feature in two directions from the sketch plane using a *closed* profile, click **Both Directions**, select **Direction 2** in the **Settings for** box, and enter the settings for **Direction 2**.

To extrude the feature in two directions from the sketch plane using an *open* profile, you indicate the settings once because **Both Directions** is selected by default.

- 9 Specify **Solid Feature** or **Thin Feature** in the **Extrude as** box.

If you are extruding the cut as a **Solid Feature**, skip ahead to Step 11. If you are extruding the cut as a **Thin Feature**, proceed to Step 10.

- 10 Select the **Thin Feature** tab at the top of the dialog box.

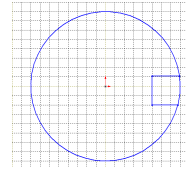
- a) Select a wall thickness **Type**. (See the descriptions on page 4-6.)

- b) Specify the **Wall Thickness**.

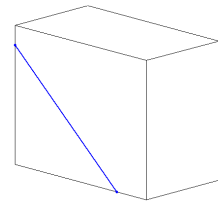
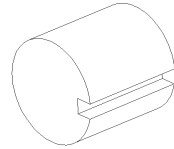
- c) To extrude in the opposite direction, click **Reverse**.

- 11 When you are satisfied with the preview, click **OK**.

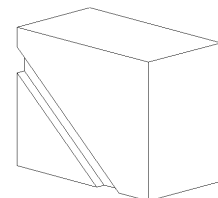
Profile for a cut



Profile extruded to make the cut



Thin feature cut from a sketched line





## Revolve

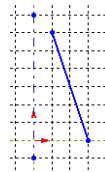


*Revolve* creates a base, boss, cut, or surface by revolving a sketch around a centerline. The default angle is 360°.

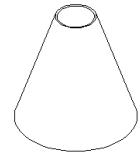
### To create a revolved base, boss, or surface:

- 1 Sketch a centerline and the profiles.
  - The sketch for a solid revolved feature can contain one or more closed, non-intersecting profiles. However, one profile must contain all of the other profiles for a base revolved feature containing multiple profiles.
  - The sketch for a thin or surface revolved feature can contain only one open, or closed, non-intersecting profile.
  - Profiles cannot cross the centerline. If the sketch contains more than one centerline, select the centerline you want to use as the axis of the revolution.
- 2 Click one of the following:
  - **Revolved Base/Boss**  on the Features toolbar, or **Insert, Base, Revolve** or **Insert, Boss, Revolve**
  - **Revolved Cut**  on the Features toolbar, or **Insert, Cut, Revolve**
  - **Insert, Surface, Revolve**
- 3 Choose a direction **Type** (**One-Direction**, **Mid-Plane**, or **Two-Direction**).
- 4 Specify the rotation **Angle**.
  - If you chose **Type** of **Mid-Plane**, the **Angle** is divided equally on both sides of the sketch plane.
  - If you chose **Type** of **Two-Direction**, specify the **Angle** for each direction.

If you used the arrow keys to specify the **Angle**, the preview shows the direction of rotation.
- 5 Click **Reverse** to rotate the feature in the opposite direction.
- 6 If creating a base or boss feature, select **Solid Feature** or **Thin Feature** in the **Revolve as** box.
- 7 For a **Thin Feature** only, click the **Thin Feature** tab.
  - Choose a direction (for the **Wall Thickness**) from the **Type** list box.
  - Specify the **Wall Thickness**.
  - Click **Reverse** to extrude the wall thickness in the opposite direction.
- 8 Click **OK**.



Sketch



Revolved feature

## Sweep


*Sweep* creates a base, boss, cut, or surface by moving a profile (section) along a path, according to these rules:

- The profile must be closed for a base or boss sweep feature; the profile may be open or closed for a surface sweep feature.
- The path may be open or closed.
- The path may be a set of sketched curves contained in one sketch, a curve, or a set of model edges.
- The start point of the path must lie on the plane of the profile.
- Neither the section, the path, nor the resulting solid can be self-intersecting.

### Simple Sweep

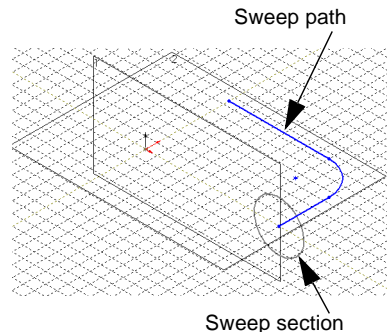
#### *To create a simple sweep:*

Sketch on multiple planes, following these general steps:

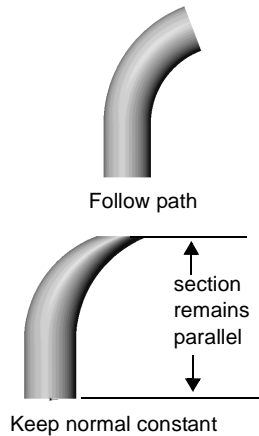
- 1 Sketch a *closed, non-intersecting* profile on one plane.
- 2 Create the path that the profile will follow. Use a sketch, existing model edges, or curves.
- 3 Click one of the following:
  - **Sweep**  on the Features toolbar, or **Insert, Base, Sweep** or **Insert, Boss, Sweep**
  - **Insert, Cut, Sweep**
  - **Insert, Surface, Sweep**

For a simple sweep, you only need to use the **Sweep** tab in the dialog box.

- 4 Click the **Sweep section** box, then select the section profile either in the model or in the FeatureManager design tree.
- 5 Click the **Sweep path** box, then select the sketch, edge, or curve that you want to use as the path either in the model or in the FeatureManager design tree.

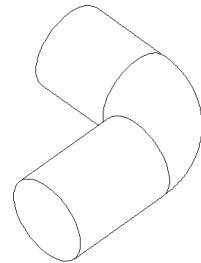


- 6 In the **Orientation/Twist control** box:
  - Select **Follow path** if you want the section to remain at the same angle with respect to the path at all times. In this example, the section is always at 90° to the path.
  - Select **Keep normal constant** if you want the section to remain parallel to the beginning section at all times.
- 7 If you picked an edge as the path, and you want to continue the sweep along all tangent edges, click **Propagate along tangent edges**.



- 8 If you want to continue the sweep profile up to the last face encountered at the ends of the path, click **Align with end faces**.
- 9 Click **OK**.

For an example of creating a simple sweep feature, refer to the tutorial, *SolidWorks 99 Tutorial*, Chapter 6.



## Sweep with Guide Curves

You can use guide curves to control the intermediate profiles as the sketch is swept along the path.

### **To create a sweep using guide curves:**

- 1 Create the guide curves, either as a sketch, a curve, or a model edge.
- 2 Create a path for the sweep. This also can be a sketch, curve, or model edge.

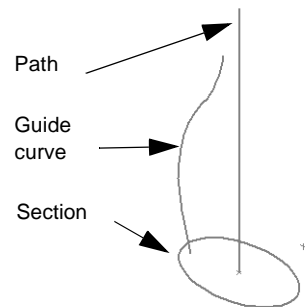
---

**NOTE:** When using guide curves to create a sweep, the path must be a single entity (line, arc, and so on) or the path segments must be *tangent* (no angled corners).

---

- 3 Sketch the sweep section.

It is recommended that you sketch the section on a plane that is normal to the end of the sweep path.





It is important to create the section *after* the path and guide curves. The intermediate profiles of the sweep are *dependent* on both the path and guide curves for their definition.

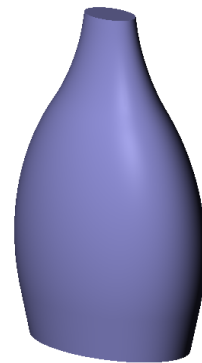
---

**NOTE:** To help you with creating the path and guide curves, you may want to create an auxiliary profile sketch first. Then, after the guide curves and path are done, derive a copy of the sketch, then underive it to break the link to the original sketch. Use this as the section in the remaining steps. See **Does it matter where I start sketching?** on page 2-48 for more information.

---

As you sketch the section, pay special attention to any relations that may be added automatically, such as **Horizontal** or **Vertical**. These relations influence the shape of the intermediate sections, and may produce results that are not what you want. If necessary, use **Display/Delete Relations**  to delete the unwanted relations. This allows the intermediate sections to twist as desired.

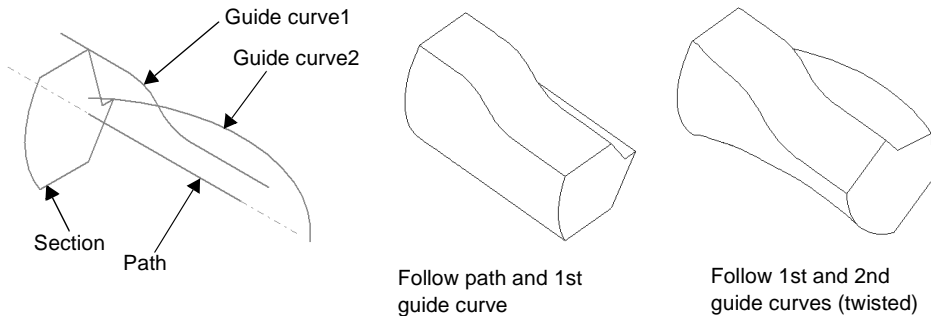
- 4 Add **Pierce** relations in the *profile* sketch, where the guide curves intersect the profile. **Pierce** relations cause the section to change size, shape, or both, along the path. It is important that the *section* be constrained to the *curves*, and not vice-versa.
- 5 Click one of the following:
  - **Sweep**  on the Features toolbar, or **Insert, Base, Sweep** or **Insert, Boss, Sweep**
  - **Insert, Cut, Sweep**
  - **Insert, Surface, Sweep**
- 6 On the **Sweep** tab, click the **Sweep section** box, then select the profile sketch in the FeatureManager design tree or in the graphics area.
- 7 Click the **Sweep path** box, then select the path (sketch, curve, or model edge).
- 8 On the **Advanced** tab, click the **Guide curve(s)** box, then select the guide curves. Use the **Up** and **Down** buttons to rearrange the order of the guide curves if needed.
- 9 Under certain conditions, these additional options for sweeps with guide curves are available on the **Advanced** tab:
  - **Maintain tangency.** If the sweep section has tangent segments, selecting this option causes the corresponding surfaces in the resulting sweep to be tangent. Faces that can be represented as a plane, cylinder, or cone are maintained. Other adjacent faces are merged, and the sections are approximated. Sketch arcs may be converted to splines.
  - **Advanced smoothing.** If the sweep section has circular or elliptical arcs, the sections are approximated, resulting in smoother surfaces. Sketch arcs may be converted to splines.
- 10 Specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Sweeps** on page 4-14.





11 On the **Sweep** tab in the **Orientation/Twist control** box, there are two additional choices besides **Follow path** and **Keep normal constant** (see page 4-11):

- **Follow path and 1st guide curve.** The twist of the intermediate sections is determined by the vector from the path to the first guide curve. The angle between the horizontal and the vector remains constant in the sketch planes of all of the intermediate sections.
- **Follow 1st and 2nd guide curves.** The twist of the intermediate sections is determined by the vector from the first to the second guide curve. The angle between the horizontal and the vector remains constant in the sketch planes of all of the intermediate sections.



12 Click **OK**.

**To view the intermediate sections in the sweep:**

You can preview the intermediate sections created along the sweep path. This option is only available for sweeps with guide curves.

- 1 Right-click the **Sweep** feature in the FeatureManager design tree and select **Edit Definition**.
- 2 On the **Advanced** tab, click **Show Intermediate Profiles**.
- 3 Use the up and down arrows beside the **Profile number** box to scroll through the profiles.

---

**NOTE:** You can activate this option before creating the sweep, if desired.

---

**Additional notes on using guide curves:**

- The path and the guide curves may differ in length. The sweep length is determined by the shortest overlapping distance.
  - If the guide curves are longer than the path, the sweep is as long as the path.
  - If the guide curves are shorter than the path, the sweep is as long as the shortest guide curve.
- Guide curves may meet at a common point, which is the apex of the swept surface.
- You can use any of the following items as a guide curve: sketched curves, model edges, or curves. See **Curve** on page 3-9 for information about creating curves.

## Start and End Tangency Options for Sweeps

With a simple sweep, end tangency is controlled by choosing either **Follow path** or **Keep normal constant** in the **Orientation/Twist control** list.

When you use guide curves to create a sweep, you have additional options for controlling the tangency at the ends of the path. You specify the **Start tangency** and **End tangency** options on the **Advanced** tab of the **Sweep** dialog box. These tangency options are only effective when you use guide curves in the sweep.

Choose from these options:

- None.** No tangency is applied.
- Path tangent.** The sweep is normal to the path at the start or end.
- Direction vector.** The sweep is tangent to a selected linear edge or axis, or to the normal of a selected plane. Click the **Direction vector** button, then click the edge, axis, or plane.
- All faces.** The sweep is tangent to the adjacent faces of existing geometry at the start or end. This selection is only available when the sweep is attached to existing geometry.

These tangency options are similar to the tangency options for lofts. Refer to the table on page 4-22 for illustrated examples.

## Loft

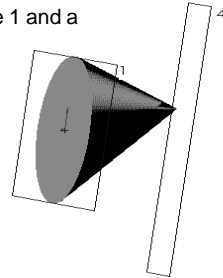


*Loft* creates a feature by making transitions between profiles. A loft can be a base, boss, cut, or surface.

You create a loft using two or more profiles. Only the first, last, or first and last profiles can be points.

For a solid loft, the first and last profiles must be model faces or faces created by split lines, planar profiles, or surfaces.

A circle on plane 1 and a point on plane 4




### Simple Loft

#### *To create a simple loft:*

1 Set up the planes needed for the profile sketches. Use existing planar faces and planes, or create new planes. The planes do not have to be parallel.

2 Sketch the profiles.

3 Click one of the following:

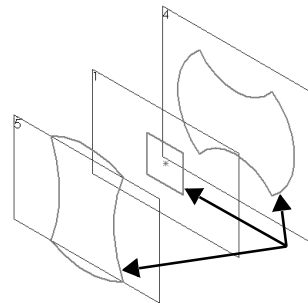
- **Loft**  on the Features toolbar, or **Insert, Base, Loft** or **Insert, Boss, Loft**
- **Insert, Cut, Loft**
- **Insert, Surface, Loft**

4 Select the profiles in order, by clicking a corresponding point on each profile.

You do not have to select the vertices precisely; the vertex closest to the selection point is used. A preview curve connecting the selected entities is displayed.

5 Examine the preview curve.

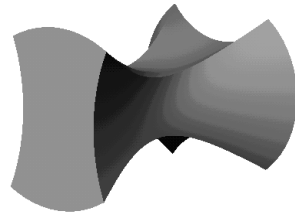
- If the preview curve looks wrong, you may have selected the sketches in the wrong order. Use the **Up** or **Down** buttons to rearrange the sketches.
- If the preview curve indicates that the wrong vertices will be connected, click the profile with the wrong vertex connected to deselect it, then click again to select a different point on the profile.
- To clear all selections and start over, right-click in the graphics area, select **Clear Selections**, and try again.



Click corresponding points

- 6 Choose from these options as needed:
  - Click **Close along loft direction** to create a closed body along the loft direction. This connects the last sketch and the first sketch automatically.
  - Click **Maintain tangency** to cause the corresponding surfaces in the resulting loft to be tangent if the corresponding lofting segments are tangent. Faces that can be represented as a plane, cylinder, or cone are maintained. Other adjacent faces are merged, and the sections are approximated. Sketch arcs may be converted to splines.
  - Click **Advanced smoothing** to obtain smoother surfaces. This option is available only if the loft sections have circular or elliptical arcs. The sections are approximated, and sketch arcs may be converted to splines.
- 7 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-21.
- 8 Click **OK**.

For examples of creating simple loft features, refer to the tutorial, *SolidWorks 99 Tutorial*, Chapter 7.




## Loft with Non-planar Profiles

### To create a loft using non-planar profiles:

- 1 Create at least one non-planar profile. The non-planar profile can be a model face or a model edge.
- 2 Set up the planes needed for the profile sketches. Use existing planes, or create new planes. The planes do not have to be parallel.
- 3 Sketch the profiles.

- 4 Click one of the following:

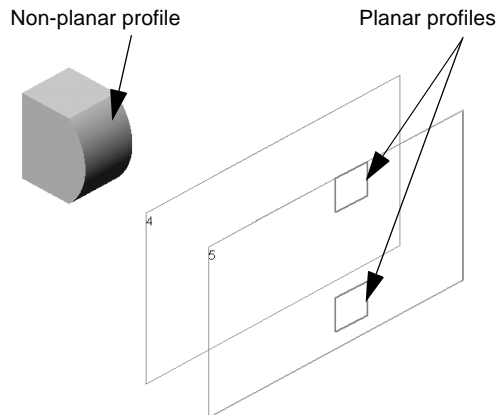
- **Loft**  on the Features toolbar, or **Insert, Boss, Loft**
- **Insert, Cut, Loft**
- **Insert, Surface, Loft**

- 5 Select the face of the non-planar profile and the other profiles in order, by clicking a corresponding point on each profile.

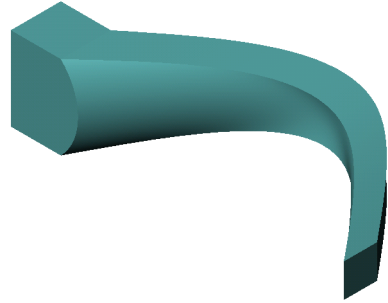
You do not have to select the vertices precisely; the vertex closest to the selection point is used.

A preview curve connecting the selected entities is displayed.

- 6 Examine the preview curve, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.




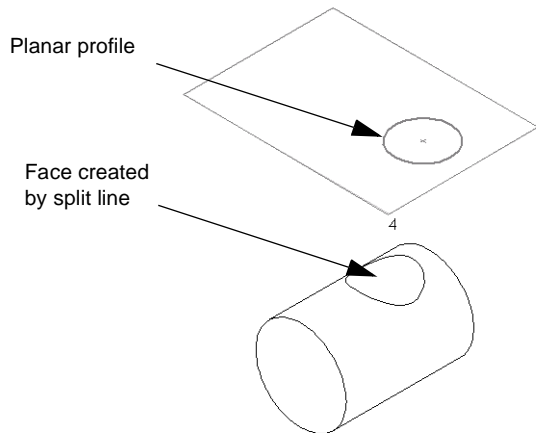
- 7 Click **Close along loft direction**, **Maintain tangency**, and **Advanced smoothing** if needed. See **Simple Loft** on page 4-15 for more information about these options.
- 8 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-21.
- 9 Click **OK**.



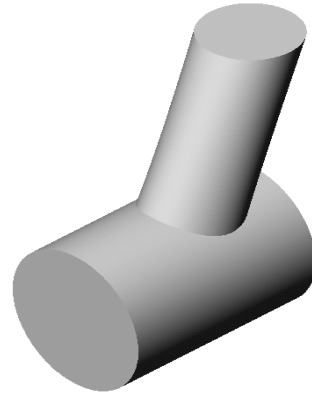
## Loft with Split Line

### *To create a loft using a split line:*

- 1 Use a split line to create a non-planar profile on a model face.  
See **Split Line** on page 3-12 for information about creating split lines.
- 2 Set up the planes needed for the profile sketches. Use existing planes, or create new planes. The planes do not have to be parallel.
- 3 Sketch the profiles.
- 4 Click one of the following:
  - **Loft**  on the Features toolbar, or **Insert, Boss, Loft**
  - **Insert, Cut, Loft**
  - **Insert, Surface, Loft**
- 5 Select the face created by the split line and the planar profiles.
- 6 Examine the preview curve, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.



- 7 Click **Close along loft direction**, **Advanced smoothing**, and **Maintain tangency** if needed. See **Simple Loft** on page 4-15 for information about these options.
- 8 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-21.
- 9 Click **OK**.

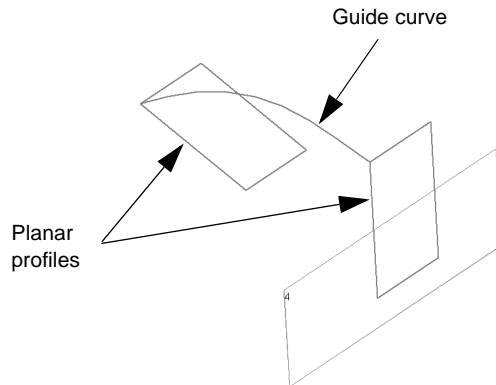


## Loft with Guide Curves


You can create a *guide curve loft* by using two or more profiles and one or more guide curves to connect the profiles. Guide curves help you control the intermediate profiles that are generated.

### To create a loft using guide curves and planar profiles:

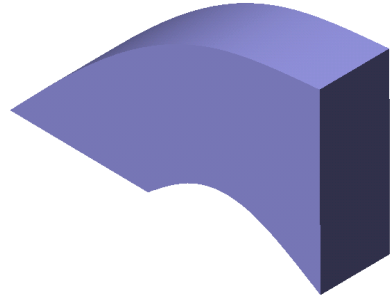
- 1 Sketch one or more guide curves.
- 2 Sketch the profiles.
- 3 Add relations between the guide curves and the profiles. You can use a combination of:
  - **Pierce** relations between the guide curves and vertices, user-defined sketch points, or both, on the profile.
  - **Coincident** relations between vertices, user-defined sketch points, or both, of the guide curves and the profiles.



These relations should be added to the profile sketches.

- 4 Click one of the following:
  - **Loft**  on the Features toolbar, or **Insert, Base, Loft** or **Insert, Boss, Loft**
  - **Insert, Cut, Loft**
  - **Insert, Surface, Loft**
- 5 Click the **Profiles** box, then select the profiles in order, in either the FeatureManager design tree or the graphics area.
- 6 Click the **Guide Curve(s)** box, then select the guide curves.
- 7 Examine the preview, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.

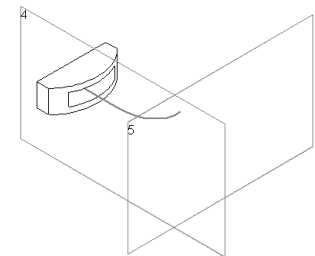
- 8 Click **Close along loft direction**, **Advanced smoothing**, and **Maintain tangency** if needed. See **Simple Loft** on page 4-15 for more information about these options.
- 9 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-21.
- 10 Click **OK**.




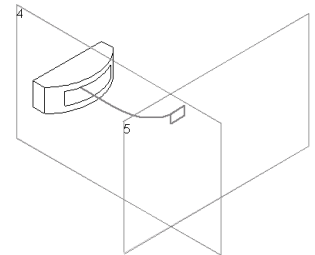
**To create a loft using guide curves and non-planar profiles:**

- 1 Use a split line to create a non-planar profile on a model face.  
See **Split Line** on page 3-12 for information about creating split lines.
- 2 Sketch one or more guide curves.
- 3 Add a **Pierce** relation between the guide curves and an edge or vertex on the non-planar face.
- 4 Sketch the remaining profiles needed to create the loft. Add **Pierce** relations between the profiles and the guide curves.

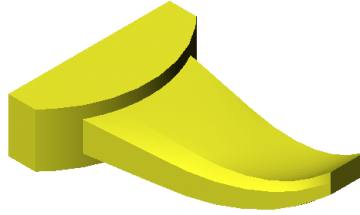
The relations should be added to the profile sketches.



- 5 Click one of the following:
  - **Loft**  on the Features toolbar, or **Insert, Boss, Loft**
  - **Insert, Cut, Loft**
  - **Insert, Surface, Loft**
- 6 Click the **Profiles** box, then select the split line face and the planar profiles in order, in either the FeatureManager design tree or the graphics area.
- 7 Click the **Guide Curve(s)** box, then select the guide curves.
- 8 Examine the preview, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.
- 9 Click **Close along loft direction**, **Advanced smoothing**, and **Maintain tangency** if needed.



- 10 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-21.
- 11 Click **OK**.




### **Notes on Creating Lofts with Guide Curves:**

- Guide curves must intersect all the profiles.
- There is no limit to the number of guide curves you may use.
- Guide curves can intersect at points.
- You can use any of the following items as a guide curve: sketched curves, model edges, or curves of any kind. See **Curve** on page 3-9 for information about creating curves.
- If a loft fails or twists, you can add a **Curve Through Reference Points** as a guide curve, selecting corresponding vertices of the profiles to create the curve. See **Curve Through Reference Points** on page 3-14.
- Guide curves can be longer than the resulting loft. The loft stops at the end of the smallest guide curve.
- You can further control the behavior of the loft by creating the same number of segments on all the guide curves. The endpoints of each segment mark corresponding points for transition of the profiles.

## Center Line Loft

You can create a loft that uses a variation of a guide curve that acts as a center line. The sketch planes of all the intermediate sections are normal to the center line. The center line may be a sketched curve, a model edge, or a curve.

### **To use a center line to guide a loft:**

- 1 Create the profiles.
- 2 Sketch a curve, or create a curve. The curve must intersect the area inside each closed profile and any split line faces.
- 3 Click one of the following, then select the profiles:
  - **Loft**  on the Features toolbar, or **Insert, Base, Loft** or **Insert, Boss, Loft**
  - **Insert, Cut, Loft**
  - **Insert, Surface, Loft**
- 4 Click the **Advanced** tab, click the **Center line** box, then click the sketched curve, model edge, or curve in the FeatureManager design tree or the graphics area.



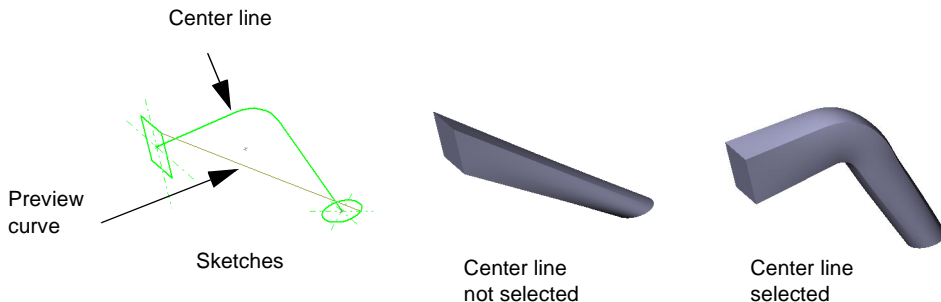
To preview the effect of the center line, click **Show Sections**. Use the slider to adjust the **Number of sections** that are displayed. To view the sections in order, click the up and down arrows beside the **Section number**.

---

**NOTE:** The number of sections you specify affects the shape of the loft. You may need to increase, or decrease, the number of sections to get the desired shape.

---

5 Click **OK**.



## Start and End Tangency Options for Lofts

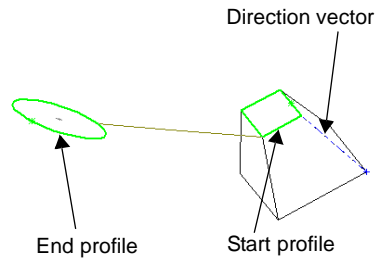
You have several options for controlling the tangency of a loft at the starting and ending profiles.

You specify the **Start tangency** and **End tangency** options on the **Advanced** tab of the **Loft** dialog box. Choose from these options:

- None.** No tangency is applied.
- Normal to profile.** The loft is normal to the sketch plane of the profile at the start or end.
- Direction vector.** The loft is tangent to a selected linear edge or axis, or to the normal of a selected plane. Click the **Direction vector** button, then click the edge, axis, or plane.
- All faces.** The loft is tangent to the adjacent faces of existing geometry at the start or end. This selection is only available when the loft is attached to existing geometry.

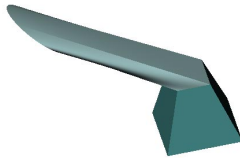
## Chapter 4 Creating Features

The examples in the following table are created from this base feature and these sketches.



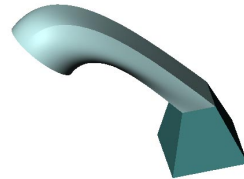
**Start tangency**  
None

**End tangency**  
None



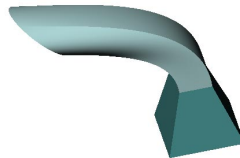
**Start tangency**  
None

**End tangency**  
Normal to profile



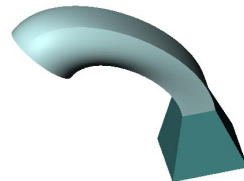
**Start tangency**  
Normal to profile

**End tangency**  
None



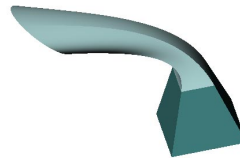
**Start tangency**  
Normal to profile

**End tangency**  
Normal to profile



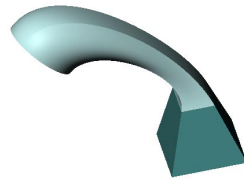
**Start tangency**  
All faces

**End tangency**  
None



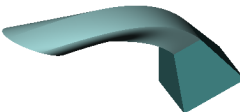
**Start tangency**  
All faces

**End tangency**  
Normal to profile



**Start tangency**  
Direction vector

**End tangency**  
None



**Start tangency**  
Direction vector

**End tangency**  
Normal to profile






*Fillet/Round* creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

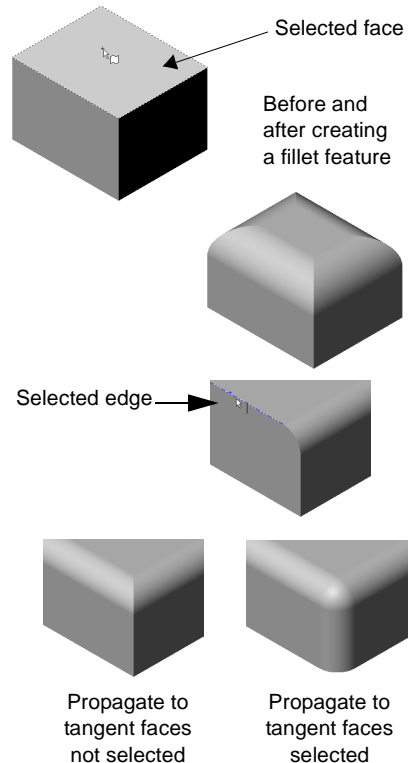
In general, it is best to follow these rules when making fillets:

- ❑ Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first.
- ❑ Add drafts before fillets. If you are creating a molded or cast part with many filleted edges and drafted surfaces, in most cases you should add the draft features before the fillets.
- ❑ Save cosmetic fillets for last. Try to add cosmetic fillets after most other geometry is in place. If you add them earlier, it takes longer to rebuild the part.
- ❑ To enable a part to rebuild more rapidly, use a single **Fillet** operation to treat several edges that require equal radius fillets. Be aware, however, that when you change the radius of that fillet, all the fillets created in the same operation change.

## Constant Radius

### *To fillet multiple model edges or faces:*


- 1 Click **Fillet**  on the Features toolbar, or **Insert, Features, Fillet/Round**.
- 2 Select the faces and edges that you want to fillet or round.
- 3 Specify the fillet **Radius**.
- 4 Select **Fillet type** of **Constant Radius**.
- 5 Verify that the correct number of edges and faces are shown in the **Items to fillet, Edge fillet items** list.
- 6 By default, the **Propagate to tangent faces** option is selected. This extends the fillet to all faces that are tangent to the selected face or edge. If you do not want the fillet to extend along tangencies, *click to clear* the check box.
- 7 Select the **Overflow Type**. See **Overflow Type** on page 4-27 for more information.
- 8 Click **OK**.



## Variable Radius

You can also create variable radius fillets. You can specify a different radius for each vertex of the edges being filleted or rounded.

### To insert variable radius fillets:

- 1 Click **Fillet**  on the Features toolbar, or **Insert, Features, Fillet/Round**.
- 2 Change the **Fillet type** to **Variable Radius**.
- 3 Select the edges that you want to fillet or round.
- 4 In the **Vertex list** under **Items to fillet**, select the vertex to which you want to assign a radius value, and type the desired radius value. Notice that the radius value appears in the **Radius** box and next to the selected vertex in the **Vertex list**. Repeat with each vertex until you have assigned all of the desired radius values.

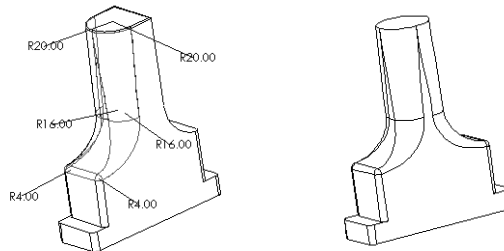
You do *not* have to assign radius values to all of the vertices in the **Vertex list**. The software calculates the radii of the vertices that are not assigned values, based on the following:

- the values of the adjacent vertices to which you have assigned radius values
- the length of edges that connect the vertices

However, you must assign a radius value to at least one vertex for each chain of edges.

You can remove an assigned radius value by selecting the vertex in the **Vertex list** and pressing the **Delete** key. Radius values calculated by the software are not shown in the **Vertex list** or in the **Radius** box.

- 5 Choose a transition type:
  - **Smooth transition** creates a fillet that changes smoothly from one radius to another when matching a fillet edge on an adjacent face.
  - **Straight transition** creates a fillet that changes from one radius to another linearly without matching edge tangency with an adjacent fillet.
- 6 Click **OK**.




To change the radius values of the fillets after they are inserted, double-click the **VarFillet** feature, then double-click and modify the dimensions displayed on the vertices in the graphics area. Radius values calculated by the software are not shown in the graphics area.

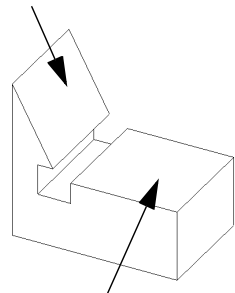
## Face Blend

You can blend non-adjacent faces with a **Face Blend** fillet.

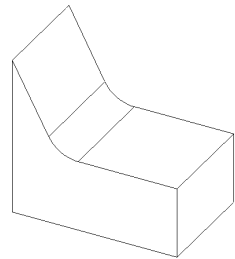
### To insert a face blend fillet:

- 1 Click **Fillet**  on the Features toolbar, or **Insert, Features, Fillet/Round**.
- 2 Change the **Fillet type** to **Face Blend**.
- 3 Click the first face or set of faces to be blended. Under **Items to fillet**, the selection is indicated in **Face set 1**.
- 4 Click the **Face set 2** box, and click the faces to blend with.
- 5 Specify the **Radius**.
- 6 If you want the blend to continue along tangencies, click **Propagate to tangent faces**.
- 7 To resolve an ambiguous selection, click **Use help point**, click the **Help point** box, then click a vertex close to the location where you want to insert the fillet.
- 8 Click **OK**.

Click this face first



Blend with this face

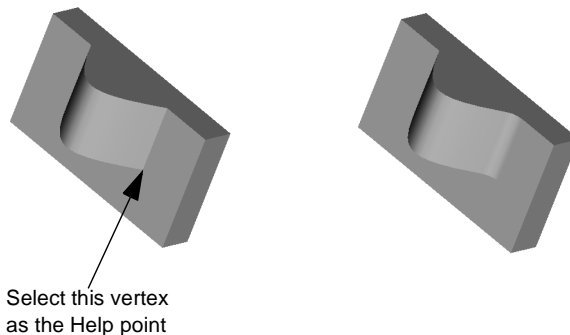


## Help Point

When creating a face blend fillet between two faces, it may not be clear where the face blend should occur. Use **Help point** to define the desired location.

In the following illustration, the location of the face fillet is ambiguous because it could be on either side of the protruding curved face.

Create a **Help point** by clicking a vertex on the side where you want the face fillet. The fillet is created at the location closer to the **Help point**.




## Face Blend with Hold Line

You can determine the radius and shape of a face blend fillet by setting a boundary, or *hold line*, for the fillet. The hold line can be an edge on the part or a split line projected on a face to be filleted.

In a fillet of this type, the radius of the fillet is driven by the distance between the hold line and the edge to be filleted.

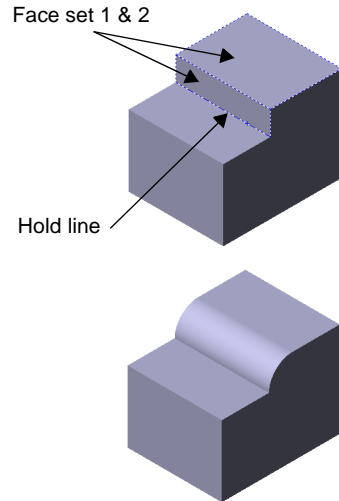
### To insert a face blend fillet using an edge:

- 1 Click **Fillet**  on the Features toolbar, or **Insert, Features, Fillet/Round**.
- 2 From the **Fillet type** box, select **Face Blend**.
- 3 Click the **Face set 1** box, then click a model face (or faces) to fillet.
- 4 Click the **Face set 2** box, then click the *adjacent* model face (or faces) to fillet.


Do not enter a radius; it is determined by the distance between the hold line and the edge to be filleted.

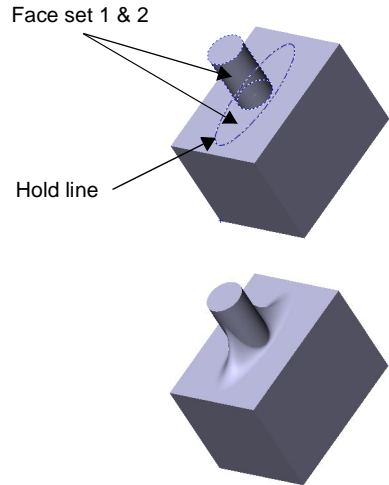
- 5 Click the **Advanced Face Fillet** tab and select the **Set fillet boundary** check box.
- 6 Click the model edge or edges that mark the end of the fillet. The names are listed in the **Hold lines** box.
- 7 Click **OK**.

The faces that share an edge with the hold lines are removed by the fillet.



### To insert a face blend fillet using a split line:

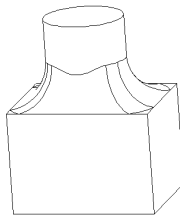
- 1 Create a split line on a model face. See **Split Line** on page 3-12 for information about creating a split line.
- 2 Click **Fillet**  on the Features toolbar, or **Insert, Features, Fillet/Round**.
- 3 Select **Insert, Features, Fillet/Round**.
- 4 Under **Fillet type**, select **Face Blend**.
- 5 Click one model face to be blended. Under **Items to fillet** the selection is indicated in **Face set 1**.
- 6 Click the **Face set 2** box, and click an *adjacent* model face.  
Do not enter a radius; it is determined by the hold line.
- 7 Click the **Advanced Face Fillet** tab, and select the **Set fillet boundary** check box.
- 8 Click the split line. The name is listed in **Hold lines** box.
- 9 Click **OK**.



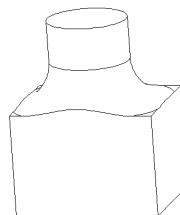
## Overflow Type

One of the options on the **Fillet Feature** dialog box is **Overflow type**. The following describes the available types and the results obtained with each type.

- **Default.** The system chooses one of the following options, depending on the geometry conditions (convexity of edges being filleted and the adjacent edges, and so on).
- **Keep Edge.** Maintains the integrity of adjacent linear edges. However, the fillet surface is broken into separate surfaces, and in many cases the top edge of the fillet may have a dip in it.
- **Keep Surface.** Uses the adjacent surface to trim the fillet. As a result, the fillet edge is continuous and smooth, but the adjacent edge is disturbed.



Keep Edge




Keep Surface

## Chamfer

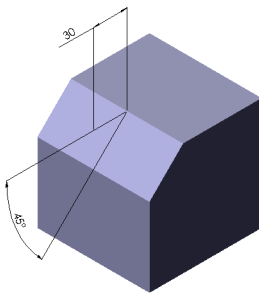
*Chamfer* creates a beveled edge on the selected edges, faces, or both.

### To create a chamfer:

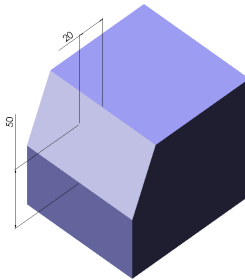
- 1 Click **Chamfer**  on the Features toolbar, or **Insert, Features, Chamfer**.
- 2 On the model, select the edges, faces, loops, or vertex to chamfer.
- 3 Look for an arrow on the selected edge; it indicates the direction of the chamfer. If the direction is not what you want, click **Flip direction**.
- 4 Examine the **Items to chamfer** list. If you selected an incorrect item, select it again to deselect, or right-click in the graphics area and select **Clear Selections**.
- 5 Select the chamfer type from the **Type** list, and specify the necessary parameters:
  - If you selected **Angle-Distance**, enter values for distance and angle.
  - If you selected **Distance-Distance**, enter values for the distances on either side of the chamfer edges, or click **Equal distance** and specify one value.
  - If you selected **Vertex-Chamfer**, enter distance values for the three chamfer edges. When you enter a distance value, the arrow moves to the corresponding chamfer edge on the model and indicates the direction and distance of the chamfer. The names of the chamfer edges are displayed in the dialog box.

– or –

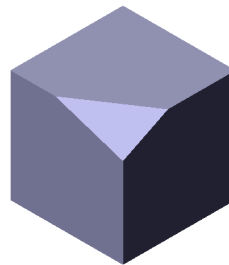
Click **Equal distance** and specify a single value. To see the direction and distance of the chamfer on an edge, select the name of the edge in the dialog box.
- 6 Click **OK**.



Angle-Distance



Distance-Distance



Vertex-Chamfer




*Draft* tapers faces using a specified angle to selected faces in the model, to make a molded part easier to remove from the mold. You can insert a draft in an existing part or draft while extruding a feature. You can draft using either a *neutral plane* or a *parting line*.

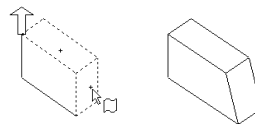
## Neutral Plane

When you draft using a neutral plane, you select a face or plane to serve as the neutral plane. The draft angle is measured perpendicular to this plane.

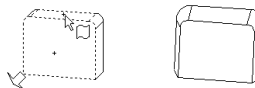
### To insert a draft angle in an existing part using a neutral plane:

- 1 Click **Draft**  on the Features toolbar, or click **Insert, Features, Draft**.
- 2 In the **Type of draft** box, select **Neutral Plane**, then click in the **Neutral plane** box and select the neutral plane in the graphics area.
- 3 Enter the **Draft angle**. If you want the draft to slant in the opposite direction, click **Reverse direction**.
- 4 Click the **Faces to draft** box, and select the faces to draft in the graphics area.
- 5 Choose the **Face propagation** type that describes how you want the draft to propagate across additional faces:

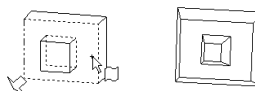
**None.** Only the selected face is drafted.



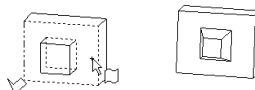
**Along Tangent.** Extend the draft to all faces that are tangent to the selected face. (The faces meet with filleted corners.)



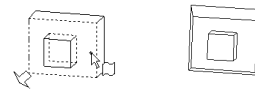
**All Faces.** Draft all faces next to the neutral plane and extruded from the neutral plane.



**Inner Faces.** Draft all faces extruded from the neutral plane.



**Outer Faces.** Draft all faces next to the neutral plane.




- 6 When you are satisfied with the results, click **OK**.

## Parting Line

The parting line option lets you draft surfaces around a parting line. The parting line can be non-planar.

To draft on a parting line, you may first divide the faces to be drafted by inserting a **Split Line** (see page 3-12), or you may use an existing model edge. Then you specify the *direction of pull*, that is, the side of the parting line from which material is removed.

### To insert a draft angle parting line:

- 1 Sketch the part to be drafted.
- 2 Insert a split line curve as described in **Split Line** on page 3-12, if desired.
- 3 Click **Draft**  on the Features toolbar, or click **Insert, Features, Draft**.
- 4 In the **Type of draft** box, select **Parting Line**.
- 5 Enter the **Draft angle**.
- 6 Click the **Direction of Pull** box, and select an edge or face in the graphics area to indicate the direction of pull.

Note the direction, and click **Reverse direction** if desired.

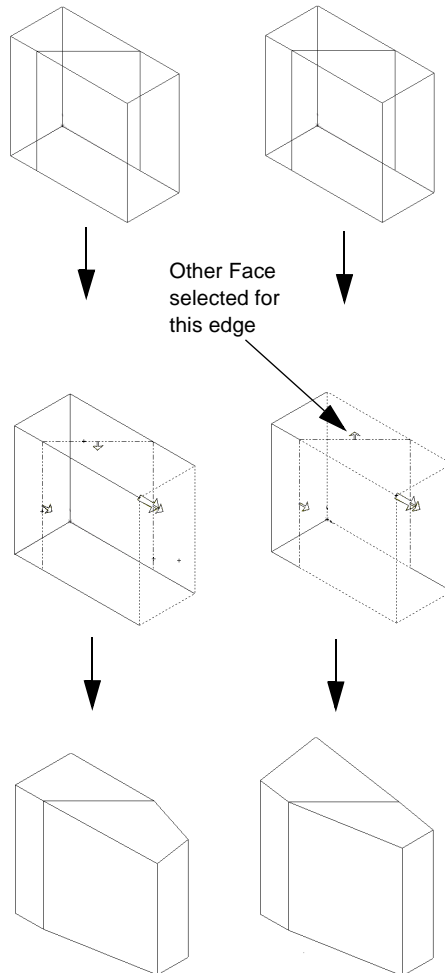
- 7 Click the **Parting lines** box, and select the parting lines in the graphics area.

Note the arrow direction. To specify a different draft direction for each segment of the parting line, click the name of the edge in the **Parting lines** box, and click **Other Face**.

- 8 Choose the **Face propagation** type:

- **None.** Draft only the selected face.
- **Along Tangent.** Extend the draft to all faces that are tangent to the selected face.


- 9 When you are satisfied with the results, click **OK**.



**To insert a draft angle using parting line with step draft:**

- 1 Sketch the part to be drafted.
- 2 Create a plane.
- 3 Create desired parting line. Make sure that:
  - At least one parting line segment is coincident to the plane on each draft face.
  - All other parting line segments are in the pull direction of the plane.
  - None of the parting line segments are perpendicular to the plane.

Insert a split line curve as described in **Split Line** on page 3-12.

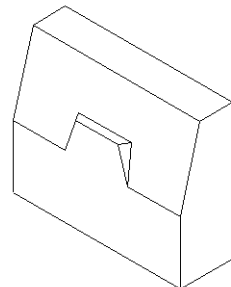
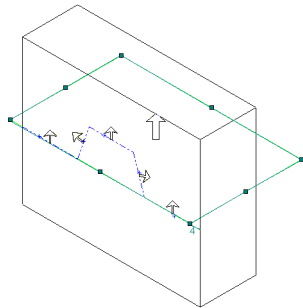
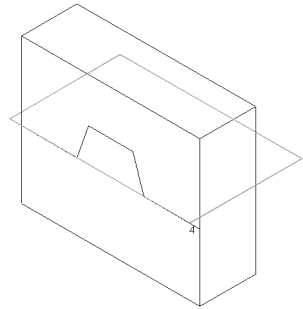
- 4 Click **Draft**  on the Features toolbar, or click **Insert, Features, Draft**.
- 5 In the **Type of draft** box, select **Parting Line**.
- 6 Click **Step draft**.
- 7 Enter the **Draft angle**.
- 8 Click the **Direction of Pull** box, and select the plane to indicate the direction of pull.

Note the arrow direction, and click **Reverse direction** if desired.

- 9 Click the **Parting lines** box, and select the parting line segments in the graphics area.



Note the arrow direction. To specify a different draft direction for each segment of the parting line, click the name of the edge in the **Parting lines** box, and click **Other Face**.

- 10 Choose the **Face propagation** type:
  - **None**. Draft only the selected face.
  - **Along Tangent**. Extend the draft to all faces that are tangent to the selected face (faces that meet with fillets or rounds.)
- 11 When you are satisfied with the results, click **OK**.




## Hole

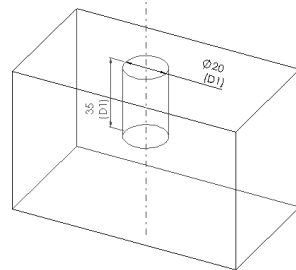
*Hole* creates various types of hole features in the model. You place a hole on a planar face, then specify its location by dimensioning it afterwards.

- Simple Hole**  places a circular hole of the depth you specify.
- Hole Wizard**  creates holes with complex profiles, such as **Counterbore** or **Countersunk**.

In general, it is best to create holes near the end of the design process. This helps you avoid inadvertently adding material inside an existing hole. Also, if creating a simple hole, use **Simple Hole**; do not use **Hole Wizard**. **Simple Hole** provides better performance than **Hole Wizard** for simple holes.

### To insert a simple hole:

- 1 Select a planar face on which to create the hole.
- 2 Click **Simple Hole**  on the Features toolbar, or click **Insert, Features, Hole, Simple**.
- 3 Select the **Type**, and specify the **Depth** or **Offset**, if necessary. For information about selecting the **Type**, see **Specifying End Conditions** on page 4-4.
- 4 Specify the **Diameter** of the hole.
- 5 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is indicated in the **Selected Items** box.
- 6 Examine the preview and click **Reverse Direction**, if necessary.
- 7 To add a draft, click **Draft While Extruding**. Enter a draft **Angle** and click **Draft Outward** if necessary.
- 8 Click **OK**.
- 9 To position the hole:
  - a) Right-click the hole feature in the model or the FeatureManager design tree, and select **Edit Sketch**.
  - b) Add dimensions to position the hole. You can also modify the hole diameter in the sketch.
  - c) Exit the sketch or click **Rebuild**.



To change the diameter, depth, or type of the hole, right-click the hole feature in the model or the FeatureManager design tree, and select **Edit Definition**. Make the necessary changes, and click **OK**.



**To insert a hole using the Hole Wizard:**

- 1 Select a planar face on which to create the hole.

---

**NOTE:** Do not select a plane; select a planar face.

---

- 2 Click **Hole Wizard**  on the Features toolbar, or click **Insert, Features, Hole, Wizard**.
- 3 Select the type of hole from the **Hole type** list (countersunk, counterbore, and so on).
- 4 In the **Section dimensions** area, double-click a dimension value to select it. Enter the new dimension.
- 5 Select the type of **End condition** from the **Type** list.
- 6 If the **Type** you specified relies on the selection of a face or vertex, click that item now in the graphics area.
- 7 Click **Next**.
- 8 Drag the centerpoint of the hole, or add dimensions to place the hole at the desired location. You can use the dimension tool  while the **Hole Placement** dialog box is displayed.
- 9 Click **Finish**.

The **Hole Placement** dialog appears.


## Scale

---

You can scale a part model about its centroid or about the model origin. The **Scale** feature only scales the *solid* geometry, for use in data export, cavities, and so on. It does not scale dimensions, sketches, or reference geometry.

A **Scale** feature is like any other feature in the FeatureManager design tree: it manipulates the solid geometry, but it does not change the definitions of features created before it was added. To temporarily restore the model to its unscaled size, you can roll back or suppress the **Scale** feature.


**To scale a model:**

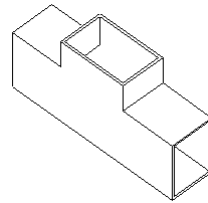
- 1 In a part document, click **Scale**  on the Features toolbar, or click **Insert, Features, Scale**.
- 2 From the **Type** list, select **About Centroid** or **About Origin**.
- 3 Specify the **Scaling factor**.
- 4 Click **OK**.

## Shell


*Shell* hollows out the part, leaving open the faces you select, and thin walls on the remaining faces. If you plan on adding fillets to the part, you should fillet the part before you shell it.

### To insert a shell:

- 1 Click **Shell**  on the Features toolbar, or click **Insert, Features, Shell**.
- 2 In the model, click the faces from which you want to remove material.  
The faces are listed in the **Faces to remove** box.
- 3 Specify the wall **Thickness**.
- 4 Click **Shell outward** if you want the shell thickness added to the outside.
- 5 Click **OK**.

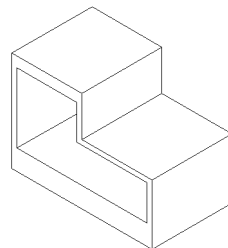
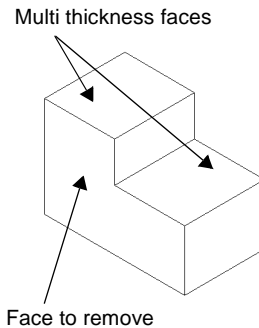


### To set a different shell thickness for each face:

- 1 Select **Shell**  on the Features toolbar, or click **Insert, Features, Shell**.
- 2 Click the faces you want removed.
- 3 Specify the default wall thickness in the **Thickness** box.
- 4 Click the **Multi thickness faces** box.
- 5 Click the walls to which you want to apply different thicknesses.

The faces are listed in the **Multi thickness faces** box.


- 6 Click each face in the **Multi thickness faces** box, and enter a **Thickness** value.
- 7 Click **Shell outward** if you want the shell thickness added to the outside.
- 8 Click **OK**.







*Rib* is a special type of extruded feature created from an open sketched contour. It adds material of a specified thickness in a specified direction between the contour and an existing part.

**To insert a rib:**

- 1 Using a plane that intersects the part, sketch the contour to be used for the rib.
- 2 Click **Rib**  on the Features toolbar, or click **Insert, Features, Rib**.
- 3 Select **Mid plane** to extrude the rib equally in both directions from the sketch plane, or **Single side** to extrude in one direction.

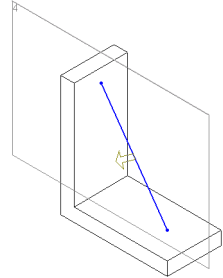
If you chose **Single side**, examine the preview and select **Reverse** if necessary.

---


**NOTE:** For better visibility, use **Isometric** and either **Hidden Lines Removed**  or **Hidden in Gray**  view mode.

---

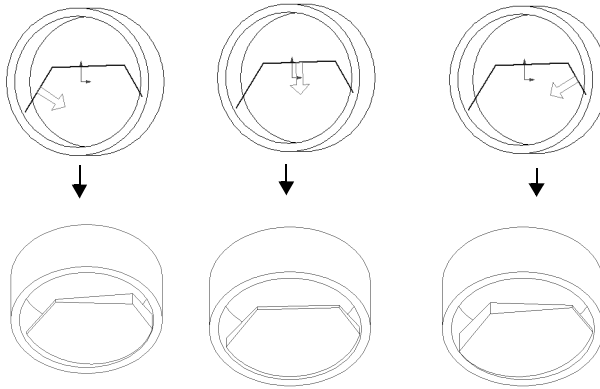
- 4 Enter the **Thickness** of the rib and click **Next**.
- 5 Note the direction of the arrow in the preview. If necessary select **Flip side of material** in the **Material direction** section, to reverse the direction in which material is added.
- 6 To add draft, select **Enable draft** in the **Draft conditions** section, and enter the **Angle**. Select **Draft outward** if necessary.
- 7 Click **Finish**.



**To insert a rib with multiple drafts:**

- 1 Sketch the part and contour lines of the rib.
- 2 Click **Rib**  on the Features toolbar, or click **Insert, Features, Rib**.
- 3 Select the **Type**, specify the **Thickness**, and click **Next**.
- 4 Select **Enable draft** in the **Draft conditions** section, and enter the **Angle**. Select **Draft outward** if necessary.

- To select the neutral plane for the draft, click **Next Reference** until the contour that represents the neutral plane is indicated. An arrow cycles around the edges to show you which edge is selected.




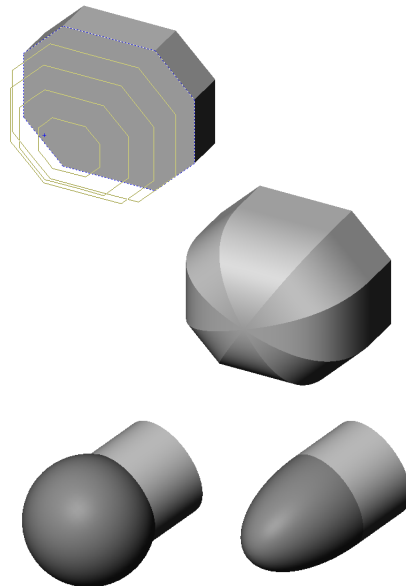
- Click **Finish**.

## Dome

You can add a *dome* feature to any planar model face.

### To create a dome on a planar face:

- Click **Dome**  on the Features toolbar, or click **Insert, Features, Dome**.
- Select a planar **Dome face** in the graphics area.
- Specify the **Height**, and observe the preview. The height is measured from the centroid of the selected face.
- Click **Reverse direction** to create a concave dome (default is convex).
- If a circular or elliptical face is selected, you can click **Elliptical dome**. This creates a dome whose shape is a half ellipsoid, with a height equal to one of the ellipsoid radii.  
A dome on a circular face may be larger in diameter than the selected face.
- Click **OK**.



Elliptical dome not selected


Elliptical dome selected



## Shape

*Shape* creates a deformed surface from a face on a model. A deformed surface is flexible, much like a membrane. It can be expanded, constrained, and tightened using the sliders on the **Controls** tab on the **Shape Feature** dialog box.

### To create a deformed surface:

- 1 Create a model with a face to shape.
- 2 Sketch one or more entities with which to constrain the shape. You can create the constraining entities directly on the face or on a plane. Valid entities are as follows:
  - Points (sketch points, endpoints, vertices, and so on)
  - Sketches
  - Edges
  - Curves
- 3 Click **Shape**  on the Features toolbar, or click **Insert, Features, Shape**. The **Shape Feature** dialog box appears.
- 4 Select the face to shape; you can only select one face.
- 5 Click the **Constrain to** box and select the entities you created to constrain the shape.

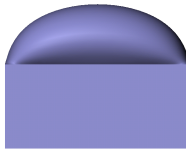
---

**NOTE:** If you constrain the shape using a vertex of a 3D sketch, you can use a feature handle to adjust the shape. For information about feature handles, see **Moving and Resizing Features** on page 5-5.

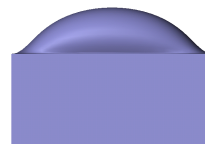
---

- 6 If desired, click to clear **Maintain boundary tangents**.

**Maintain boundary tangents**  
not selected



**Maintain boundary tangents**  
selected



- 7 Click **Preview**. Rotate the part and examine the shape from different angles.

---

**NOTE:** If the selected face does not change shape, use the sliders on the **Controls** tab to adjust the shape.

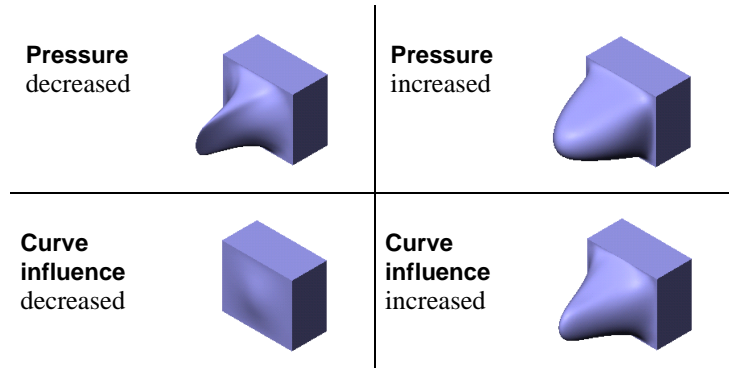
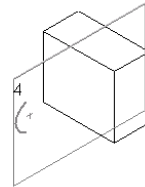
---

8 Click the **Controls** tab to adjust the shape.

Adjust the sliders and observe the preview. *Experiment* until you are satisfied with the results.

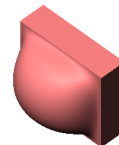
- **Pressure.** Deflates or inflates the shape.
- **Curve influence.** Moves the shape away from or toward the constraining entities. Note that the **Curve influence** control is unavailable if **Constrain to** is not selected on the **Shape Feature** tab.

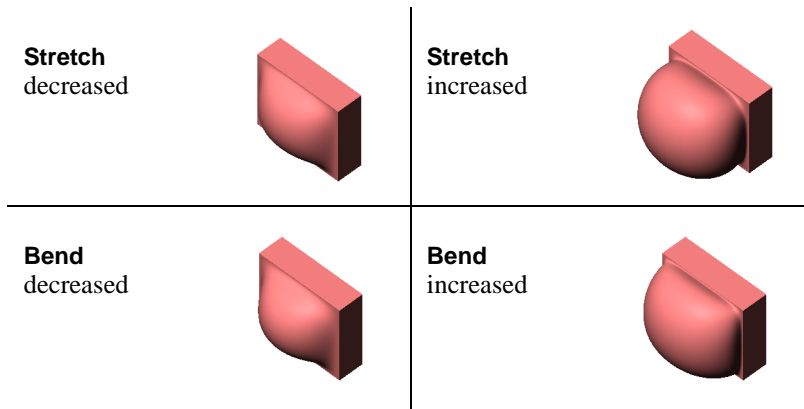
The following four illustrations are derived from this base feature and sketch, and show how you can create different shapes by adjusting the sliders.



- **Stretch.** Adjusts the degree of stretching.
- **Bend.** Adjusts the degree of bending.

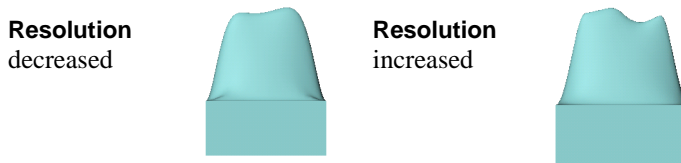
The following four illustrations show the effects of adjusting the **Stretch** and **Bend** sliders. All four illustrations are derived from this base feature, which was shaped by increasing **Pressure** on the front face.







- **Resolution.** Adjusts the precision of the shape by changing how many points on the face are deformed.

Higher **Resolution** often smoothens the flat areas and sharpens the shape near the constraining entities. Lower **Resolution** provides better performance and a coarser shape; however, some details may be omitted.



After you make the first adjustment to the shape, the undo button  is displayed on the **Controls** tab. Click the undo button  to undo the previous adjustment. You can undo an unlimited number of adjustments.

- 9 When you are satisfied with the shape of the deformed surface, click **OK**.

To change the shape of a deformed surface, right-click the **Shape Feature** in the FeatureManager design tree and select **Edit Definition**. Then repeat Steps 8 and 9.

## Pattern and Mirror

---

*Pattern* repeats the selected features in a linear or circular array.

*Mirror* copies the selected features or all features, mirroring them about the selected plane or face.

- For a **Linear Pattern**, you select the features, then specify the direction, the linear spacing, and the total number of instances.
- For a **Circular Pattern**, you select the features and an edge or axis as the center of rotation, then specify:
  - The total number of instances and the angular spacing between instances.  
– or –
  - The total number of instances and the total angle in which to create the pattern.
- For a **Mirror Feature**, you select the features to copy and a plane about which to mirror them.
- To **Mirror All**, you select a planar face on the model, then mirror the entire model (base and all other features) about the selected face.


You can also create patterns of patterns, and mirrored copies of patterns.

For information about using a pattern of components in an assembly, see **Adding a Component Pattern** on page 7-11.

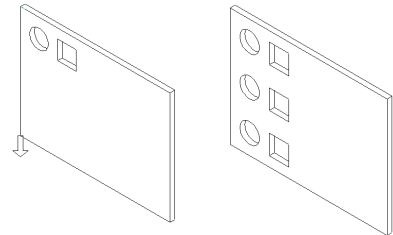
## Linear Pattern

You can use a linear pattern to quickly create multiple copies of one or more features in one or two directions.

### To create a linear pattern:

- 1 Create a base part and on the base part, create one or more cut, hole, or boss features that you want to repeat.
- 2 Click **Linear Pattern**  on the Features toolbar, or click **Insert, Pattern/Mirror, Linear Pattern**.
- 3 With the **Items to copy** box selected, click the features to pattern.

- 4 Click the **Direction selected** box, then click a model edge or dimension to indicate the pattern direction or driving dimension for the **First Direction**. Notice the direction arrow on the model.



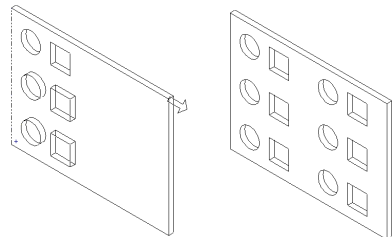
- 5 Click **Reverse direction** if the arrow points in the wrong direction.
- 6 Specify the values for **Spacing** and **Total instances** of the pattern.

- **Spacing** is the distance from a point on one feature to the corresponding point on the adjacent copy (*not* edge-to-edge spacing).
- **Total Instances** includes the original features on which the pattern is based.

If you change either value using the arrow keys, a preview of the resulting pattern is automatically displayed. If you change either value by typing in a new value, click the **Preview** button to update the preview.

- 7 Click **Vary sketch** if you want the pattern to change as it is repeated. See **Controlling and Modifying Patterns** on page 4-46 for more information about using **Vary sketch**.
- 8 Click **Geometry pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-46.
- 9 Click **OK** to create the pattern in one direction, or continue with Step 10 to create the pattern in two directions.

- 10 Select **Second Direction**, then click a different model edge to indicate this direction. Again, you may need to click **Reverse direction**.



- 11 Set the values for **Spacing** and **Total instances** of the pattern in the **Second Direction** and examine the preview.

- 12 Click **OK**.

## Circular Pattern

You use a circular pattern to create multiple copies of one or more features in rotation about an axis.


### **To create a circular pattern:**

- 1 Create a base part and on the base part, create one or more cut, hole, or boss features that you want to repeat.
- 2 Create an axis or use an existing linear edge or axis around which to pattern the feature.

---

**NOTE:** For any circular object, there is a temporary axis. If axes are not displayed, click **View, Axes** (for user-defined axes) or **View, Temporary Axes** (for axes created implicitly by the model).

---

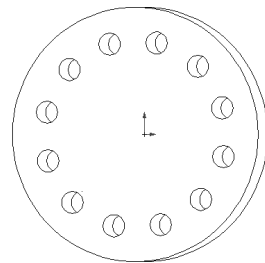
- 3 Click **Circular Pattern**  on the Features toolbar, or click **Insert, Pattern/Mirror, Circular Pattern**.
- 4 With the **Items to copy** box selected, click the features to pattern.
- 5 Click the **Direction selected** box, then select an axis, model edge, or angular dimension to define the pattern.
- 6 Specify the **Total instances** of the feature. (**Total instances** includes the original features on which the pattern is based.)
- 7 Specify the distance between each instance in the **Spacing** box.

– or –

Select the **Equal spacing** check box. Then specify the total angle in which to create the pattern in the **Total angle** box.

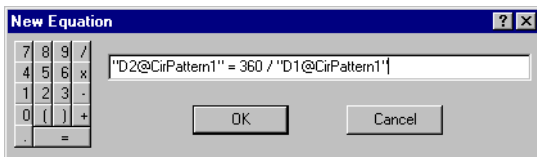


If you change **Total instances**, **Spacing**, or **Total angle** using the arrow keys, a preview of the resulting pattern is automatically displayed. If you change the values by typing in new values, click the **Preview** button to update the preview.

- 8 Click **Reverse direction** if you want to create the pattern in a counterclockwise direction. Otherwise, the pattern goes in a clockwise direction.
- 9 Click **Vary sketch** if you want the pattern to change as it is repeated. (Refer to **Vary Sketch** on page 4-47 for more information.)
- 10 Click **Geometry pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-46.
- 11 Click **OK**.



You can use an equation to calculate the **Spacing** angle used in a circular pattern.

**To use an equation to control the spacing angle in a circular pattern:**


- 1 Create a circular pattern as described in the **Circular Pattern** section. Leave the default values in the **Spacing** and **Total Instances** boxes. It is not important what numbers you use at this time.
  - 2 In the FeatureManager design tree, double-click the circular pattern feature. Two values appear in the model: **Spacing** angle and **Total instances**.
  - 3 Click **Tools, Equations**, then click **Add** in the **Equations** dialog box.
  - 4 Click the **Spacing** angle dimension in the model. The dimension name is entered in the **New Equation** dialog box.
  - 5 Using the calculator keypad in the dialog box, click **= 360 /**.
  - 6 Click the **Total Instances** value in the model. Its full name is added to the end of the equation.
  - 7 Click **OK** to complete the equation, and click **OK** again to close the **Equations** dialog box. Note the **Equations** folder  in the FeatureManager design tree.
- If you change the number of **Total instances** in the pattern, the **Spacing** angle is recalculated, so that the instances are always equally spaced around the center.
- 8 Click **Rebuild**  or **Edit, Rebuild**.

For more information about using equations, see **Equations** on page 5-9.

## Mirror Feature

**Mirror Feature** creates a copy of one or more features, mirrored about a plane or planar face. You can either use an existing plane or create a new one. If you modify the original feature, the mirrored copy is updated to reflect the changes.

### To mirror a feature or features:

- 1 Click **Mirror Feature**  on the Features toolbar, or click **Insert, Pattern/Mirror, Mirror Feature**.
- 2 With the **Mirror plane** box selected, click a plane or face.
- 3 With the **Features to mirror** box selected, click a one or more features on the model or in the FeatureManager design tree.

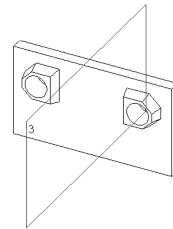
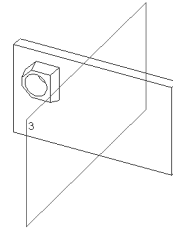
A preview of the mirrored features is displayed.

- 4 Click **Geometry pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-46.
- 5 Click **OK**.

---

**NOTE:** You can also mirror a feature about a face that is perpendicular to the sketch plane. Mirroring will be about the selected face, not about the feature itself.

---



## Mirror Pattern

### To create a mirrored copy of a pattern:

- 1 Click **Insert, Pattern/Mirror, Mirror Feature**.
- 2 With the **Mirror plane** box selected, select a plane about which to mirror the pattern.
- 3 With the **Features to mirror** box selected, click the *pattern* to be mirrored. You only need to select *one instance* of one feature of the pattern in the model.

A preview of the mirrored pattern is displayed.

- 4 Click **Geometry pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-46.
- 5 Click **OK**.



## Mirror All

To create a part that is symmetrical about a planar face, you can use **Mirror All**. You build one half of the part, then mirror the entire model all at once. Any changes you make to the original half are reflected in the mirrored half.

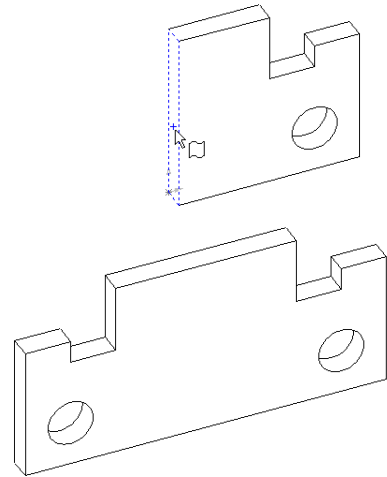
### *To mirror a part around a planar face:*

- 1 Establish a plane corresponding to the plane of symmetry of the complete part. This can be an existing plane or one you create.
- 2 Create the features for one half of the part on one side of this plane.
- 3 Click **Insert, Pattern/Mirror, Mirror All**.
- 4 Select the face of the part half on the plane of symmetry.

The name of the face appears in the **Mirror Plane** box.

- 5 Click **OK**.

A mirror image of the original part half is joined to the part at the selected face to make a complete, symmetrical part.



## Pattern of Patterns

### *To create a pattern of patterns:*

- 1 Select **Insert, Pattern/Mirror**, then select either **Linear** or **Circular** pattern.
- 2 Select the pattern feature in the FeatureManager design tree, or select a face on a pattern instance in the model.
- 3 Proceed as described in **Linear Pattern** on page 4-41 or **Circular Pattern** on page 4-42.

## Controlling and Modifying Patterns

You can control and modify feature patterns in the following ways:

- Use the **Geometry pattern** option to make exact patterned and mirrored copies of features, and to improve system performance when using patterned and mirrored features.
- Use the **Vary sketch** option to adjust the profile of the patterned feature based on its dimensions and relations to other features.
- Use a mathematical **Equation** to calculate values that define the pattern.
- Use **Edit Seed Feature** to modify the original feature of a pattern or mirrored feature based on a single feature.
- Use **Delete Pattern Instances** to remove selected instances in a pattern.

Pattern instances inherit the feature color of the original feature as long as:

- the pattern is based on one feature.
- the color of the pattern or any faces on any pattern instances has not been changed.

For information about changing the colors of faces and features, see **Part Color and Appearance** on page 5-2.

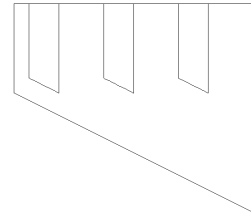
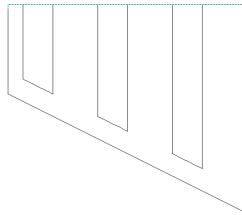
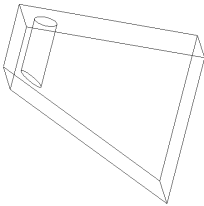
### Geometry Pattern

You can make mirrored or patterned copies of features using the **Geometry pattern** option. **Geometry pattern** (a check box on the pattern and mirror dialog boxes) copies the faces and edges of the geometry being patterned or mirrored. Any references to other model geometry (such as **Up to Surface**, **Offset from Surface**) used to create the original feature are ignored (not solved).

The **Geometry pattern** check box also improves system performance when using complex patterned or mirrored features.

In the following example, the cut is extruded using the **Offset from Surface** end condition. When **Geometry pattern** is *not selected*, the copies retain this design parameter. The end of the cut is offset the same distance from the selected face in each instance.

When **Geometry pattern** is *selected*, the copies are exact copies of the faces and edges of the original feature. The end condition is ignored, and the distance from the end of the cut to the selected face is not recalculated.



Offset From Surface end condition

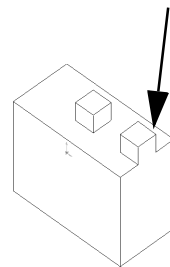
Geometry Pattern not selected

Geometry Pattern selected

**Notes on Geometry pattern:**

- You cannot use **Geometry pattern** with any feature that has a face in common with another face on the model.
- Geometry pattern** is automatically selected when creating patterns on **Shape** and **Dome** features.

Feature with face in common with another face



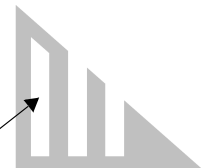
**Vary Sketch**

Use the **Vary sketch** option if you want the pattern to change its dimensions as it is repeated. For example, you may want to maintain a specific distance between the edges of the base part and the patterned features.

In this example, the first feature (the cut feature) is patterned three times.

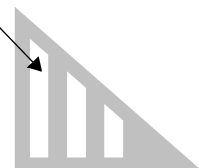
- When **Vary sketch** is *not selected*, the pattern remains the same regardless of the defining geometry.
- When **Vary sketch** is *selected*, the pattern maintains its relationship to the sloping edge, based on the dimensions and constraints of the first instance of the pattern.

Linear pattern without Vary sketch



first feature

Linear pattern with Vary sketch



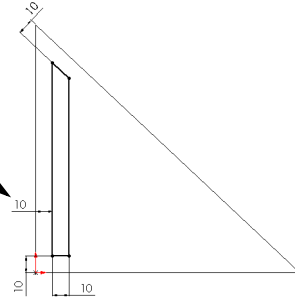
### To create a variable pattern:

- 1 Create a sketch for the first feature in the pattern, observing the following recommendations:
  - The feature sketch must be constrained to the boundary that defines the variation of the pattern instances. For example, in the illustrated pattern, the angled top edge of the first feature is parallel and dimensioned to the angled edge on the base part.
  - The feature sketch should be *fully defined*.

- 2 Double-click one of the feature's dimensions and click the arrow to change the dimension.


This gives you a preview of the way this dimension will drive the feature when you create the pattern. Try a different dimension for a different result.

Double-click this dimension

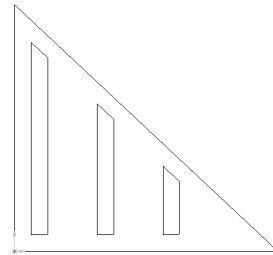


- 3 When you are satisfied that the dimensions are correct and the pattern will repeat as you wish, click **Insert, Cut, Extrude** (or **Insert, Boss, Extrude**) to create the first feature.

- 4 In the FeatureManager design tree, select the feature to repeat.

- 5 Click **Linear Pattern**  or **Insert, Pattern/Mirror, Linear Pattern**.

- 6 Click the **Direction selected** box, then click the dimension that you want to use to drive the pattern.



- 7 Select the **Vary sketch** check box.

- 8 Specify the values for the **Spacing** and **Total instances** of the pattern.

- 9 Click **OK** to create the pattern.

### Edit Seed Feature

You can edit the definition of the original feature of a linear pattern, a circular pattern, a mirrored feature, and a mirrored pattern. The pattern or mirrored feature must be based on one feature.

### To edit the definition of the original feature:

- 1 Right-click a pattern or mirrored instance, and select **Edit Seed Feature**.
- 2 Change the desired parameters.
- 3 Click **OK**.

## Delete Instance

### *To delete an instance of a feature from a pattern, or an entire pattern:*

- 1 Select the pattern instances on the model that you want to delete.
- 2 Press the **Delete** key.
- 3 In the dialog box, select one of the following:
  - **Delete Pattern Instances.** Only the selected instances are deleted. The instance locations are displayed in the **Instances Deleted** box.
  - **Delete Pattern Feature.** The complete pattern is deleted, not including the original feature.
- 4 Click **OK** to delete, or click **Cancel** to exit the dialog box.

### *To restore deleted pattern instances:*

- 1 Right-click the pattern feature in the FeatureManager design tree, and select **Edit Definition**.
- 2 In the **Instances deleted** box, select the numbers corresponding to the deleted instances to restore, and press the **Delete** key.
- 3 Click **OK** to restore the deleted instances.

## Delete Feature from a Multi-feature Pattern

### *To delete a feature from a multi-feature pattern:*

- 1 Select the feature in the FeatureManager design tree that you want to delete.
- 2 Press the **Delete** key.
- 3 In the **Confirm Delete** dialog box, select the **Also delete absorbed features** check box, if available.
- 4 Click **Yes** to delete the feature from the pattern and to close the dialog box.  
The feature is deleted from the pattern.

## Surfaces

---

Surfaces are a type of geometry that can be used to create solid features. You can create surfaces by these methods:

- Insert a planar surface from a sketch or from a set of closed edges that lie on a plane
- Extrude, revolve, sweep, or loft from sketches
- Offset from existing faces or surfaces
- Import a file (For information about importing surfaces see Chapter 14, “Importing and Exporting Files.”)

**You can use surfaces in the following ways:**

- Select edges and vertices and use as a sweep guide curve and path.
- Create a solid or cut feature by thickening a surface.
- Extrude a solid or cut feature with the end condition **Up to Surface** or **Offset from Surface**.
- Create a solid feature by thickening surfaces that have been knit into a closed volume.

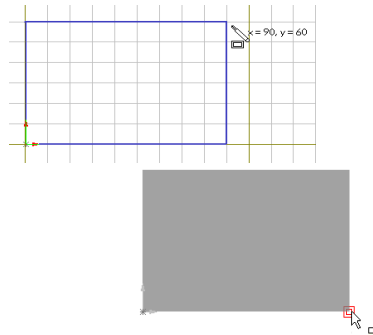
### Creating Surfaces

The methods you can use to create surfaces are similar to those you use to create solid features in the model. You can extrude, revolve, sweep, or loft a surface from sketches, knit, or offset from existing model face or surface. A surface may have either an open or closed profile.

**To create a bounded planar surface:**

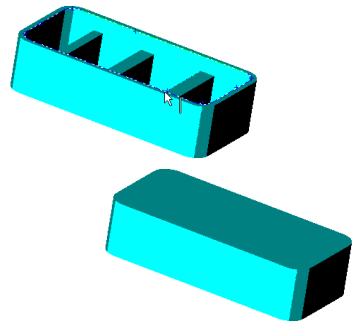
- Create a non-intersecting, single contour, closed sketch and select **Insert, Surface, Planar**. You can select edges and vertices, the same as a model.

To make changes to this planar surface, you can edit the sketch.



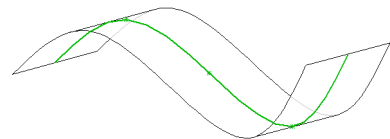
- Select a set of closed edges in a part or assembly, and select **Insert, Surface, Planar**. (All edges in the set must be on the same plane.)

You can change this planar surface by right-clicking the planar surface and selecting **Edit Definition**.



**To extrude a surface:**

- 1 Sketch the profile of the surface.
- 2 Click **Insert, Surface, Extrude**.
- 3 In the **Extrude Feature** dialog box, on the **End Condition** tab, choose the **Type**, and specify the **Depth**.
- 4 Examine the preview. If the offset is in the wrong direction, click **Reverse Direction**.
- 5 Click **OK**.

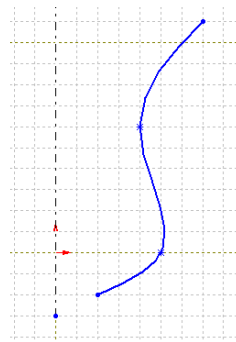


Extruded surface (MidPlane type) from a sketched spline

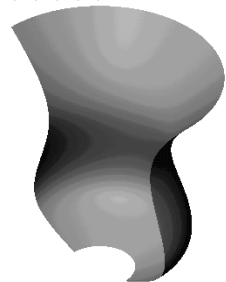
See **Extrude** on page 4-2 for more information about extruded features.

**To revolve a surface:**

- 1 Sketch a profile and a centerline around which to revolve the profile.
- 2 Click **Insert, Surface, Revolve**.
- 3 Type the value for **Degrees**, and click **Reverse Direction** if necessary.
- 4 Click **OK**.



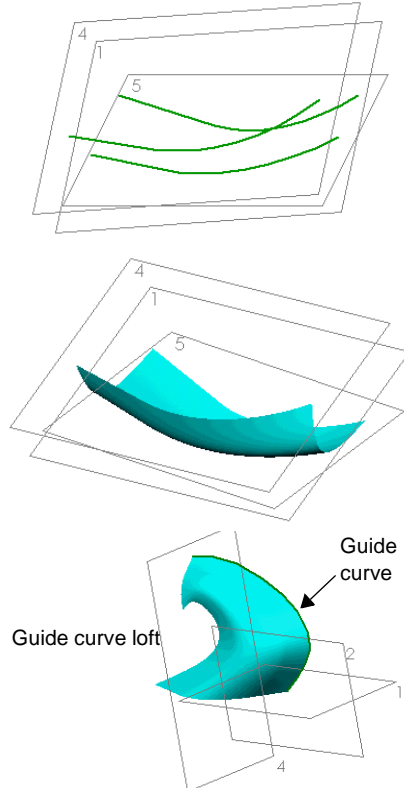
See **Revolve** on page 4-9 for more information about revolved features.



**To loft a surface:**

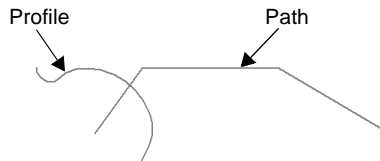
- 1 Create a plane for each profile section of the loft. (The planes do not need to be parallel.)
- 2 Sketch the section profiles and guide curves (if any are needed) on the planes. Close each sketch after the sketch is complete.
- 3 Click **Insert, Surface, Loft**.
- 4 Click the profiles in the order in which you want them to be connected. Select corresponding segments on each profile; the vertex closest to the selection point is used to connect the profiles.
- 5 If you are using guide curves, click the **Guide Curve** box in the **Loft Surface** dialog, then select the guide curves.
- 6 Choose from the options **Maintain Tangency**, **Close along Loft Direction**, and **Advanced Smoothing** as desired.
- 7 Click **OK**.

See **Loft** on page 4-15 for more information about lofted features.



**To sweep a surface:**

- 1 Create planes as needed for sketching the sweep profile, sweep path, and the guide curves (if any are needed).
- 2 Sketch the sweep profile and path on the planes that you created.




---

**NOTE:** You may also sketch the sweep path on a model face or use a model edge for the path.

---

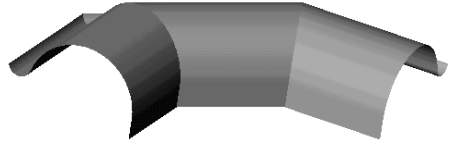
- 3 If you are using one or more guide curves, create coincident or pierce relations between the guide curves and the profile.
- 4 Click **Insert, Surface, Sweep**.
- 5 Click the **Sweep Section** box, then click the profile sketch.
- 6 Click the **Sweep Path** box, then click the path sketch or model edge, if that is what you are using for a path.
- 7 If you are using guide curves, click the **Advanced** tab. Click the **Guide Curve** box, then click the sketch, curve, or model edge for the guide curves.



- 8 Select options as desired under **Orientation/Twist Control, Advanced Smoothing, and Maintain Tangency.**

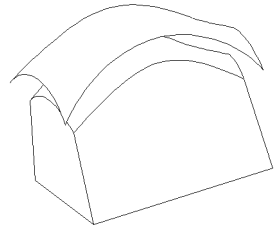
- 9 Click **OK.**

See **Sweep** on page 4-10 for more information about creating swept features.



#### **To offset a surface:**

- 1 Click **Insert, Surface, Offset.**
- 2 In the **Offset Surface** dialog, specify the **Offset** value.
- 3 Select one or more model surfaces from which to create an offset.
- 4 Examine the preview, and click **Reverse** if necessary.
- 5 Click **OK.**



Offset Surface from a lofted model surface

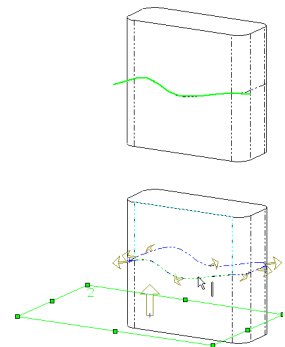
### **Radiate Surface**

You can create surfaces by radiating a parting line, an edge, or a set of contiguous edges inwards or outwards, and parallel to a selected plane.

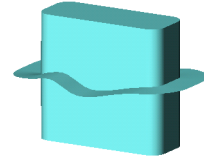
You can use radiate surfaces to cut a mold into two or more parts. For more information about using radiate surface to cut a mold, see **Mold with Complex Parting Lines** on page 7-21.

#### **To create a surface by radiating a parting line:**

- 1 Sketch a curve on the face of a part.
- 2 Project the curve as a **Split Line**. (See **Split Line** on page 3-12 for more information.)
- 3 Click **Insert, Surface, Radiate.**
- 4 Click the **Reference plane** box and select a plane that is parallel to the direction in which you want the surface to radiate.
- 5 Set the width of the surface in the **Radiate distance** box.
- 6 Click the split line as the **Edges to radiate**. Note the arrow direction. To specify a different direction, click **Reverse direction**.



- 7 Select **Propagate along tangent faces** if you want the surface to continue around the tangent faces of the part.
- 8 Click **OK**.



## Knit Surfaces

Use the **Knit Surface** function to combine two or more surfaces or faces into one. The edges of the surfaces or faces must be adjacent and not overlapping. They do not need to be on the same plane.

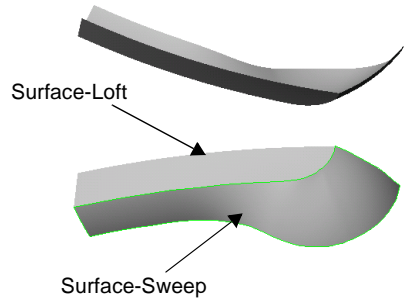
This is an illustration of two views of a pair of adjacent surfaces: a lofted surface and a swept surface.

### *To knit surfaces:*

Click **Insert, Surface, Knit**, and select the surfaces or faces in the graphics area. You can also select surfaces in the FeatureManager design tree.

The result is a single surface listed in the FeatureManager design tree as

**Surface-Knit<n>**. There is no difference in the appearance of the surfaces after knitting.



### *To thicken knit surfaces:*

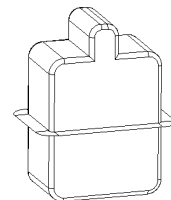
- 1 Click **Insert, Boss, Thicken**.
- 2 Specify a value in the **Thickness** box.
- 3 Select the side to thicken, or select **Thicken both sides**.
- 4 Click the **Surface to thicken** box, and select the **Surface-Knit** icon in the FeatureManager design tree.
- 5 Click **OK**.



### *To use the Seed face option:*

To knit surfaces using the **Seed** option, you must use **Radiate Surface**.

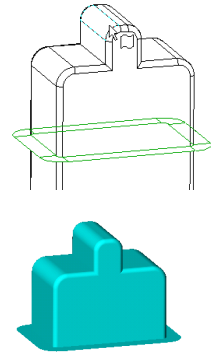
- 1 Project a split line onto the model and use it to create a radiate surface that completely encompasses the model. (See **Radiate Surface** on page 4-53.)
- 2 Click **Insert, Surface, Knit**.
- 3 In the **Faces and surfaces** box, select the radiate surface.



- 4 Click the **Seed face** box, and then click any face on the model that you want to knit into a surface.

The **Seed** surface propagates the face selection to all the model faces until it reaches the radiate surface. It then knits all of those selected faces into a surface.

(For this illustration, the **Section View** tool was used to separate the model from the surface.)

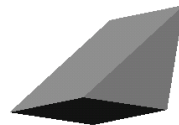
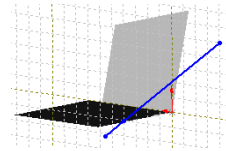


***To fill a volume with a loft between two surfaces:***

You can create a solid body by lofting between two surfaces.

- 1 Create adjacent surfaces.
- 2 Open a new sketch, sketch a guide curve, and close the sketch.
- 3 Click **Insert, Boss, Loft**.
- 4 Click the **Profiles** box and select the surfaces.
- 5 Click the **Guide Curve(s)** box and select the guide curve sketch.

The resulting lofted body is solid.

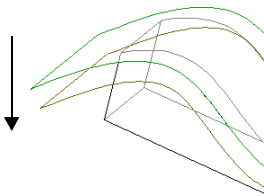
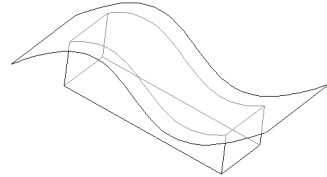


## Using Surfaces to Create Features

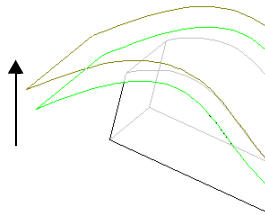
A surface has no thickness. You can thicken a surface to create model geometry, or to cut into existing geometry. You can also use a zero-thickness surface to cut through a model, similar to cutting with a parting line.

### To create a solid feature by thickening a surface:

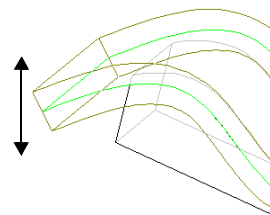
- 1 Create a surface as described in the previous section, or import an IGES surface.
- 2 Click **Insert, Base, Thicken** (if there are no features in the part) or **Insert, Boss, Thicken**.
- 3 Select the surface.
- 4 In the **Thicken Feature** dialog, specify the desired **Thickness** for the feature.
- 5 Specify the sides you want to thicken. Examine the preview to see the effect.



Thicken Side 1



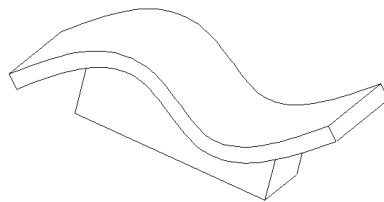
Thicken Side 2



Thicken Both Sides

If you are creating a boss, the thickened surface *must* intersect the existing part.

- 6 Click **OK**.



**To cut a part with a surface:**

1 Create a surface in the part model, or import one.

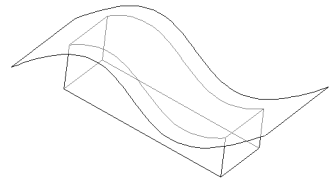
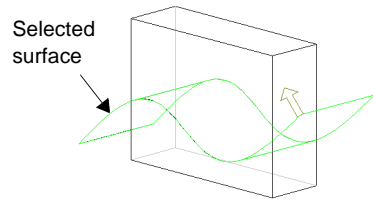
2 Select the surface.

3 Click **Insert, Cut, With Surface**.

Note the preview arrow, indicating which side of the part will be cut away.

4 In the **Surface Cut** dialog box, click **Flip the Side to Cut Away** to reverse the direction of the cut, if necessary.

5 Click **OK**.

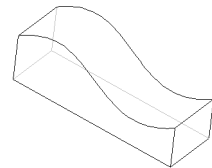


6 To hide the surface, if desired, right-click the **Surface** feature in the FeatureManager design tree and select **Hide**.

---

**NOTE:** You can also cut a part with a surface by using the **Up To Surface** option when extruding a cut.

---



**To cut a part by thickening a surface:**

1 Create a surface in the part model, or import one.

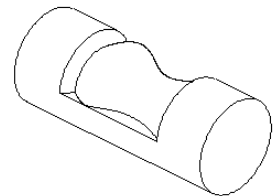
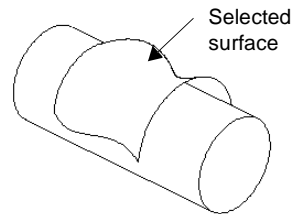
2 Select the surface.

3 Click **Insert, Cut, Thicken**.

4 In the **Thicken Feature** dialog, type the desired thickness for the feature.

5 Specify which side to thicken.

6 Click **OK**.



## Working with Parts

---

The 3D part is the basic building block of the SolidWorks mechanical design software. This chapter describes parts and some ways to work with them, including:

- Editing, moving, and copying features
- Using equations
- Dependency editing using Rollback, Suppress, and Unsuppress
- Displaying a section view of the model
- Annotating parts
- Specifying lighting characteristics and material properties

## Feature Editing

---

In a part document, you can perform the following types of feature editing operations:

- Edit the definition, the sketch, or the properties of a feature.
- View the parent/child relationships of a feature.
- Use the feature handles to move and resize features.
- Control the access to selected dimensions.
- Change the order in which features are reconstructed when the part is rebuilt.  
See **Drag and Drop Features** on page 1-21.
- Roll back the part to the state it was in before a selected feature was added.  
See **Rollback** on page 5-15.

To perform some of these operations, you use the icons in the FeatureManager design tree, where the features are listed in the order in which they are created. For more information, see **The FeatureManager Design Tree** on page 1-18.

### Edit Definition

You can edit the definition of a feature to change its parameters. For example, you can edit the depth of an extruded feature, or the selected edges for a fillet.


#### *To edit the definition of a feature:*

- 1 Click a feature either in the graphics area or in the FeatureManager design tree, then click **Edit, Definition**, or right-click the feature and select **Edit Definition**. Depending on the selected feature type, the appropriate dialog box appears.
- 2 Edit the definition in the dialog box by specifying new values or options.
- 3 Click **Apply** or **OK** to accept the changes; click **Cancel** to discard the changes.

### Part Color and Appearance

You can change the color of an entire part, selected features (including surfaces or curves) or selected model faces. You can also edit the properties that define the way the material reacts to light.

#### *To change the color of a part, a face, or a feature:*

- 1 Click **Edit Color**  on the Standard toolbar.
- 2 Select the item to change:
  - To change the color of the *entire part*, click the part name at the top of the FeatureManager design tree.
  - To change the color of a *feature*, click a face on the feature, or click the feature name in the FeatureManager design tree. Features can include not only solid features of the model, but also surfaces and curves.
  - To change the color of a *face*, click the model face.

- 3 Select a color from the color palette, or specify a custom color.
- 4 If you selected a face, specify whether you want to apply the change to only the selected **Face**, or to the entire **Feature**.  
If you selected a feature in the FeatureManager design tree, the color is automatically applied to the whole feature.
- 5 Click **Apply**.  
If the new color is not acceptable, you can select a different color, and click **Apply** again. The **Color** dialog box remains open, so you can change the color of other items.  
To restore a face to the color of the feature it belongs to, or to restore a feature to the color of the part, or to restore the part to the default color, select the item and click **Remove**. (Face colors override the color of the feature, feature colors override the color of the part.)
- 6 When you are done changing colors, click **OK** to dismiss the dialog box.

***To change the appearance of a part:***

- 1 Click **Tools, Options, Color**.
- 2 Under **System**, select **Shading**.
- 3 To change the color, click the **Edit** button, then choose a color from the palette, or click **Define Custom Colors** to specify a custom color.  
Click **OK** to close the **Color** dialog box.
- 4 To change the way the part reacts to light, click the **Advanced** button, then use the sliders to change the display properties.
  - **Ambient**. Light reflected and scattered by other objects
  - **Diffuse**. Light scattered equally in all directions on the surface
  - **Specularity**. Ability of a surface to exhibit highlights
  - **Shininess**. A glossy, reflective surface or a dull, matte surface
  - **Transparency**. Ability to pass light through the surface
  - **Emission**. Ability to project light from the surfaceClick **OK** to close the **Material Properties** dialog box.
- 5 Make sure that **Apply To** is set to **Active Document**, then click **OK** to close the **Options** dialog box.

See also **Lighting** on page 5-27.



## Properties of Features and Faces

Feature properties include name, color, and suppression state. Face properties include name and color.

### *To edit feature properties:*

- 1 In the FeatureManager design tree, select one or more features, then click **Edit, Properties**, or right-click the features and select **Feature Properties**.

You can also select one or more features in the graphics area, then right-click and select **Feature Properties**. (To select multiple features, hold down the **Ctrl** key while selecting.)

The **Feature Properties** dialog box appears.

- 2 Enter a new **Name** if desired.

- 3 To suppress the feature, click the **Suppressed** check box.

Suppressing a feature removes it from the model temporarily, without deleting it. This can be used to simplify the model, or to create part configurations. See **Suppressing Features** on page 5-13 and **Creating a Configuration Manually** on page 8-2.

- 4 To change the appearance of the feature, click **Color**. The **Entity Property** dialog box appears.

- To change the feature color, click **Change Color**. Select a color from the color palette or define a custom color.
- To change material properties such as shininess and transparency, click **Advanced**. (See the previous section for more information)
- To restore the feature to the color of the part, click **Use Part Color**.

- 5 Click **OK**.

### *To edit face properties:*

- 1 Right-click one or more faces and select **Face Properties**. The **Entity Property** dialog box appears.

- 2 Enter a face **Name** if desired. (Face names are used under certain conditions when replacing components in an assembly. See **Replacing a Component in an Assembly** on page 6-12.)

- 3 To change the face color, click **Change Color**. Select a color from the color palette or define a custom color.

To change material properties such as shininess and transparency, click **Advanced**.

To restore the face to the color of the feature it belongs to, click **Use Part Color**. (Face colors override colors specified for the *feature*.)

- 4 Click **OK**.

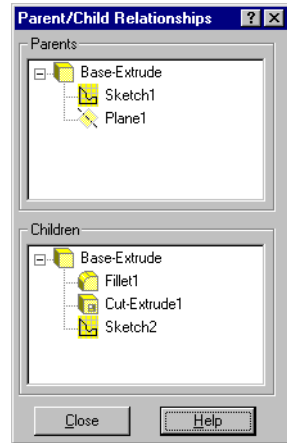
## Parent/Child Relationships

When features are built upon other features, their existence depends upon the existence of the previously built feature. The new feature is called a *child* feature. For example, a hole is the child of the feature in which it is cut.

A *parent* feature is an existing feature upon which others depend. For example, a boss is the parent feature to a fillet that rounds its edges.

### To view Parent/Child relationships:

- 1 In the FeatureManager design tree or in the graphics area, right-click the feature whose relationships you want to see.
- 2 Select **Parent/Child** from the menu. You can only *view* the relationships; you cannot edit them.




## Moving and Resizing Features

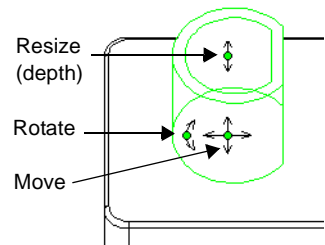
You can use the **Move/size features** tool  on the Features toolbar in two ways:


- To display and manipulate *feature handles*. Feature handles (available on extruded and revolved features, domes, and some shape features) let you move, rotate, and resize the features quickly and easily.
- To perform *dynamic feature editing*. When you modify the entities in a sketch that is used to create a feature, you can see a dynamic preview of the resulting feature.

### Feature Handles

#### To display and use the feature handles:

- 1 Click the **Move/size features** tool  on the Features toolbar. (If this button is not displayed on your toolbar, click **View, Toolbars, Customize**, and follow the instructions on the **Commands** tab to add it to a toolbar.) This button stays enabled while you continue working in the part.
- 2 Click the feature you want to work with, either in the FeatureManager design tree or in the graphics area.
- 3 To rotate or resize a feature, drag the appropriate handle. (Domes have only the resize handle.)



- 4 To move a feature, you can either:
  - Drag the feature to a new location. If the feature has locating dimensions or relations that restrict its movement, you are given the choice either to delete them, to leave them dangling, or to cancel the operation.
  - Hold down the **Alt** key as you drag to maintain the dimensions and relations on the feature. The feature is allowed to move only within unspecified degrees of freedom.
- 5 When you are finished, click  again to remove the handle display.

You can also move a feature by holding the **Shift** key and dragging one of its faces. The behavior is the same as using the move handles, and you can use the **Alt** key the same way.

When you *drag a feature with the move handle*, any child feature moves with its parent. When you **Shift**-drag, an attempt is made to maintain the relative position of any child feature, based on the child's dimensions and relations. However, in some cases (for example, when the child feature is under defined, or dimensioned to something other than its parent), the child may not move with the parent as expected.

To move more than one feature at a time, hold down the **Ctrl** key as you select the features. This is useful in the case of under defined child features.


### Dynamic Feature Editing

You can see a dynamic feature preview when you drag the entities of a sketch, either with or without opening the sketch itself. This is a fast, easy way to edit any feature that is based on one or more sketches.

To control the behavior of the preview, click **Tools, Options, Performance**, and select or clear the **Dynamic update on sketch drag** check box.

- When selected, the preview updates if you pause momentarily while dragging a sketch entity, or when you modify a dimension. If the model is displayed in **Shaded** mode, the preview is transparent.
- When cleared, the preview updates when you release the pointer after dragging.

#### ***To display a dynamic preview without opening a sketch:***


- 1 Click the **Move/size features** tool  on the Features toolbar.
- 2 Double-click the feature you want to edit.

The sketch is highlighted in the graphics area. If more than one sketch is used to create the feature (for example, if the selected feature is a loft or a sweep), all the sketches used in the feature are highlighted.

Notice that the model is *not rolled back* when you edit a feature this way.

- 3 Drag the sketch entities as needed. As you modify the sketch, the preview shows you what the resulting feature will look like.
- 4 When you are satisfied with the feature, click an empty area of the document window or press **Esc** to clear the selected sketch entity.

***To display a dynamic preview while editing a sketch:***

- 1 Click the **Move/size features** tool  on the Features toolbar.
- 2 Double-click a sketch. The sketch is opened for editing, and the model is rolled back to the selected feature.
- 3 Modify the sketch by dragging the sketch entities, changing the dimensions, adding relations, and so on. As you modify the sketch, the preview shows you what the resulting feature will look like.
- 4 Exit the sketch to update the feature.


For detailed information about sketch editing, see Chapter 2, “Sketching.”

## **Copying Features**



***To copy a feature:***

- 1 Hold down the **Ctrl** key, then drag the feature where needed. You can drop the feature on the same face, or on a different one. To copy more than one feature at a time, hold down the **Ctrl** key as you select the features.

If the feature has locating dimensions or relations that restrict its movement, you are given the choice either to delete them, to leave them dangling, or to cancel the operation.

- 2 Either delete or re-attach any dangling dimensions. Edit the sketch, select the dangling dimension, and drag the red handle to a new attachment point. As you drag, the  pointer is displayed until you are on an edge or vertex that is suitable to re-attach the dimension.

***To copy features from one part to another part:***

Tile the windows, then drag features from one window to another, or use the **Copy**  and **Paste**  tools on the Standard toolbar.

## Controlling Access to Dimensions

For some types of parts, such as sheet metal forming tools and library feature parts, you may want to prevent other users from changing certain dimensions.

When you drag one of these items into a part document (from the Feature Palette window, or from Windows Explorer, or from a hyperlink in Internet Explorer), the dimensions are displayed, and they are available for editing. You can control the access to each dimension, to prevent other users from making unwanted changes when they use these items. See **The Feature Palette Window** on page 11-6 for more information.

You can also use this function to reduce the clutter of unnecessary dimensions in the display.

### **To control dimension access:**

- 1 Right-click the part icon at the top of the FeatureManager design tree, and select **Edit Dimension Access**.

In the **Dimension Access** dialog box, there are two lists: **User dimensions** (accessible) and **Internal dimensions** (inaccessible).

- 2 Specify the dimensions for each list:
  - To move a dimension from one list to the other, click its name, then click the left or right arrow button. As you click each name, the dimension is displayed in the model.
  - To move *all* the dimensions at once, click the double arrow button.
- 3 Click **Finish**.




When you insert an item with restricted dimension access into a part, only the **User dimensions** are displayed and available for editing. The **Internal dimensions** are not displayed, and they are not available for editing.

## Equations



---

You can use equations to define mathematical relations between parameters or dimensions. Equations can include sketch dimensions and feature parameters (such as the depth of an extruded feature, or the instance count in a pattern). Equations can also be used to relate the dimensions of different parts in an assembly, or the values (distances or angles) used in mates.




### To add an equation:

- 1 Click **Tools, Equations**, or click **Equations**  on the Tools toolbar. (To display the Tools toolbar, click **Tools, Customize**.)
- 2 In the **Equations** dialog box, click **Add**.
- 3 Enter the equation by clicking the dimensions or parameters in the model, and the numbers and operators in the calculator keypad, either in the **New Equation** dialog box or on the keyboard. The maximum number of characters allowed in an equation is 512. Equations are solved left to right (the expression on the left is driven by the expression on the right), *in the order in which they appear in the list*. Therefore, the order in which you enter equations can affect the resulting part.
- 4 Click **OK**. The equation is listed in the **Equations** dialog box, and the resulting value is displayed in the **Evaluates To** column.
- 5 If you want to add another equation, click **Add** again; otherwise, click **OK** to close the **Equations** dialog box. Notice the **Equations** folder  in the FeatureManager design tree.
- 6 Click **Rebuild**  to apply any changes caused by solving the equation.




### To add more equations:

- 1 Click **Tools, Equations** or click **Equations** , then click **Add**.  
– or –  
Right-click the **Equations** folder , and select **Add Equation**.
- 2 Repeat Steps 3 through 6 from the previous procedure.

### To edit an equation:



- 1 Click **Tools, Equations** or click **Equations** , then click **Edit All**.  
– or –  
Right-click the **Equations** folder , and select **Edit Equation**.
- 2 In the **Edit Equations** dialog box, edit the text of the equations as needed, and click **OK**. The revised equation is displayed in the **Equations** dialog box. You can also cut and paste equations here to change the order, if necessary.
- 3 Click **OK**, then click **Rebuild** .

**To delete an equation:**

- 1 Click **Tools, Equations** or click **Equations** .  
– or –  
Right-click the **Equations** folder , and select **Delete Equation**.
- 2 Click the equation number or the status marker (see the next section) for the equation to delete, and click the **Delete** button in the **Equations** dialog box.
- 3 Click **OK**, then click **Rebuild** .

## Equation Status

When you delete a feature or a dimension that is used in an equation, the software asks you if you want to delete the equation also. Because the order of solving equations is important, you may prefer to keep the equation, and edit it later.

If you keep the equation, it has **Not Solved** status, and is marked with a red exclamation point  in the **Equations** dialog box. Equations with **Solved** status are marked with green check marks .

## Comments in Equations

You can add comments to equations, to document your design intent. You can also use comment syntax to prevent an equation from being evaluated.

**To add a comment to an equation:**

In the **New Equation** dialog box, type a single quote (‘) at the end of the equation, then enter the comment. You can also add comments in the **Edit Equations** dialog box. Anything after the single quote is ignored when the equation is evaluated.

For example: "D2@Sketch1" = "D1@Sketch1" / 2 ‘height is 1/2 width

If you want to temporarily prevent an equation from being evaluated, but you do not want to delete it, place a single quote *before* the equation. The equation is then interpreted as a comment, and it is ignored.

## Operators and Functions

The following operators and functions are supported:

+	(plus sign)	addition
-	(minus sign)	subtraction
*	(asterisk)	multiplication
/	(forward slash)	division
^	(caret)	exponentiation

You can also use **pi** in equations. Its value is calculated to the number of decimal places specified for the document.


**Table 5-1. Functions available in equations**


Function	Name	Notes
$\sin(\alpha)$	sine	$\alpha$ is angle expressed in radians
$\cos(\alpha)$	cosine	$\alpha$ is angle expressed in radians
$\tan(\alpha)$	tangent	$\alpha$ is angle expressed in radians
$\text{atn}(a)$	inverse tangent	$a$ is angle expressed in radians
$\text{abs}(a)$	absolute value	returns the absolute value of $a$
$\text{exp}(n)$	exponential	returns $e$ raised to the power of $n$
$\log(a)$	logarithmic	returns the natural log of $a$ to the base $e$
$\text{sqr}(a)$	square root	returns the square root of $a$
$\text{int}(a)$	integer	returns $a$ as an integer
$\text{sgn}(a)$	sign	returns the sign of $a$

## Measuring Sizes and Distances

You can measure the size of items, and the distance and angle between lines, points, surfaces, and planes in sketches, part or assembly models, or drawings.

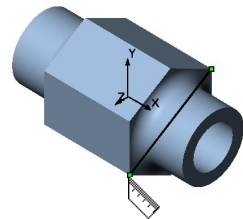
### *To measure the size of, or the distance between, entities:*

- 1 Click **Tools, Measure**, or click **Measure**  on the Tools toolbar. (To display the Tools toolbar, click **Tools, Customize**.)

To keep the dialog box in place while you are working, click the pushpin .

- 2 Click the entities to measure.

- If you select a single entity, the size of the entity (the length of an edge, the area of a face, and so on) is displayed in the **Measurements** box.
- If you select multiple entities, the distance between them is displayed, as well as many other dimensions and conditions. These include parallelism, perpendicularity, or angle between items, length of arcs and chords, perimeter, radius, and so on.



- 3 In the **Projection on** area, click **Screen** to measure the projection to the screen; click **Plane/Face** to measure the projection to a selected plane or planar face.

- 4 To select different items to measure:

- Click a selected item again to deselect it, then click a new item.
- Click an item in the **Selected items** list and press the **Delete** key.
- Right-click in the graphics area, and select **Clear Selections** to start again.

The measurements update dynamically when you change selections.



---

**NOTE:** To temporarily turn off the measuring function, right-click in the graphics area, and choose **Select** from the shortcut menu. To turn the measuring function on again, click inside the **Measure** dialog box.

---

- 5 If you have defined a coordinate system, you can display the results based on that system, instead of the using the document's default coordinate system. Select the name from the **Output coordinate system** list.

See **Creating a Coordinate System** on page 3-7 for information about creating a user-defined coordinate system.

- 6 Click **Options** if you want to change the way the results are displayed. The **Measurement Options** dialog box appears.

To display the results using **Scientific notation**, select the check box.

To display the results using different units than the units specified for the active document, select one of these systems of units:

- **View measurement settings.** This is a *temporary* system of units. Changing the units here does not affect the units specified for the document, or the system default units (for new documents). You can specify the **Length unit**, **Angular unit**, and other options as needed (decimal places, display of fractions, and so on).
- **View global settings.** Displays the units specified for the active document.
- **View system defaults.** Displays the system default units for new documents.

You can view the settings for the active document and the system defaults, but you cannot change them here. (To change them, click **Tools, Options, Grid/Units.**)

When you click **OK**, the values in the **Measurements** box update automatically to reflect the changed units.

- 7 Click **Close** to close the **Measure** dialog box.

## Dependency Editing

---

The Features toolbar contains the following tools to suppress and unsuppress features of the model.



**Suppress**



**Unsuppress**



**Unsuppress with Dependents**

If the buttons do not appear on your toolbar, you can add them. Click **Tools, Customize**, click the **Commands** tab, then follow the instructions in the dialog box.

### Suppressing Features

You can suppress a feature so you can work on the model with the selected feature temporarily removed from the model. Suppressing a feature not only removes it from the display, but also from any calculations in which it may be involved. Rebuilding complex models occurs faster when features with a large amount of detail are suppressed.

Suppression is governed by the parent/child relationships of the features. See **Parent/Child Relationships** on page 5-5 for more information.


Examine these illustrations:




<b>Tooth Cut</b>	Unsuppressed	Suppressed	Unsuppressed
<b>Tooth Copies</b>	Unsuppressed	Suppressed	Suppressed

- In the first example, the **Tooth Cut** feature (the parent) and the **Tooth Copies** feature (the child, a circular pattern of **Tooth Cut** features) are unsuppressed.
- In the second example, **Tooth Cut** (the parent) is suppressed, causing **Tooth Copies** (the child) to be suppressed also (the child is *dependent* on the parent).
- In the third example, unsuppressing **Tooth Cut** (the parent) does not automatically unsuppress **Tooth Copies** (the child). However, unsuppressing a child feature causes its parent feature to be unsuppressed.

**To suppress a feature:**

- 1 Select the feature in the FeatureManager design tree, or select a face of the feature in the graphics area. To select multiple features, hold down **Ctrl** as you select.
- 2 Click **Edit, Suppress**, or **Suppress**  on the Features toolbar.  
– or –
  - 1 In the FeatureManager design tree, right-click a feature, and select **Properties**.
  - 2 In the **Feature Properties** dialog box, select the **Suppressed** check box, and click **OK**.  
The selected feature is removed from the model (but *not* deleted). The feature disappears from the model view and is shown in gray in the FeatureManager design tree.

**To unsuppress a feature:**



- 1 Select the suppressed feature in the FeatureManager design tree.
- 2 Click **Edit, Unsuppress**, or **Unsuppress**  on the Features toolbar.  
– or –
  - 1 In the FeatureManager design tree, right-click a feature, and select **Properties**.
  - 2 In the **Feature Properties** dialog box, clear the **Suppressed** check box, and click **OK**.

---

**NOTE:** While a feature is suppressed, the only way to select the feature to unsuppress it is through the FeatureManager design tree.

---

**To unsuppress a feature and its dependents:**

- 1 Select the suppressed parent feature in the FeatureManager design tree.
- 2 Click **Edit, Unsuppress with Dependents**, or **Unsuppress with Dependents**  on the Features toolbar.  
The selected feature and any features that are dependent on it are returned to the model.  
– or –
  - 1 Select the child feature in the FeatureManager design tree.
  - 2 Click **Edit, Unsuppress**, or **Unsuppress**  on the Features toolbar.  
The selected feature and its parent feature are returned to the model.

## Rollback

*Rollback* reverts the model to the state it was in before a selected feature was created. When a part is rolled back, new features are inserted at the point in the FeatureManager design tree where you rolled back, instead of at the bottom of the tree. Use this method to change a part at an earlier point of its development.

### **To roll back with the rollback bar:**

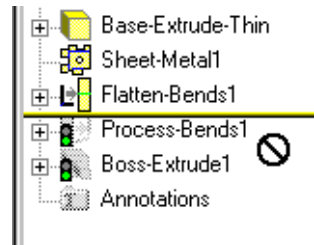
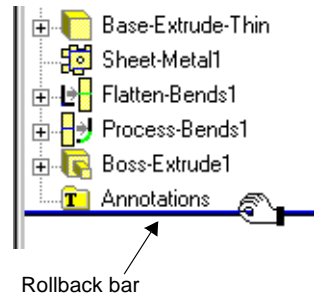
- 1 Point to the rollback bar in the FeatureManager design tree. The pointer changes to a hand.
- 2 Click to select the rollback bar. The bar changes color from yellow to blue.
- 3 Drag the rollback bar up the FeatureManager design tree until it is above the feature you want rolled back.

– or –

Use the up and down arrow keys on the keyboard to move the rollback bar up or down. (Select the **Arrow key navigation** check box in **Tools, Options, General** to turn on this functionality.)

The rolled back icons are gray, and cannot be selected.

You can also select a feature in the FeatureManager design tree, and click **Edit, Rollback**. The rollback bar moves above the selected feature.



### **To roll forward:**

Move the rollback bar down the FeatureManager design tree. You can:

- Roll forward all the way by dragging to the bottom of the tree.
- Roll forward a few steps at a time, by dragging the rollback bar to the desired location.
- Roll forward one feature at a time. Select the rollback bar, then use the up and down arrow keys to rebuild the features one by one. (Select the **Arrow key navigation** check box in **Tools, Options, General** to enable this functionality.)

## Derived Parts

---

You can create a new part directly from an existing part. The new part, called a *derived part*, has the original part as its first feature, and it is linked to that part by means of an *external reference*. This means that any changes you make to the original part are reflected in the derived part.

When a part has an external reference, its name in the FeatureManager design tree is followed by an arrow ->. To see the name, location, and status of the externally referenced document, right-click the derived part, and select **List External References**.

There are three types of derived parts: **Base Part**, **Mirror Part**, and **Derived Component Part**.

### Base Part

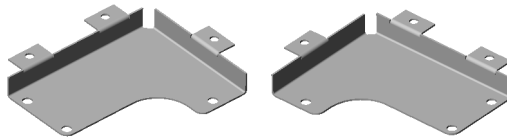
**Base Part** uses an existing part as the base feature for the derived part.

*To create a derived part from an existing part document:*

- 1 Open a new part document.
- 2 Click **Insert, Base Part**.
- 3 Browse to the original part document (.sldprt) that you want to use as a base feature, and click **Open** (or double-click the part document). The derived part is opened in the new part window.

### Mirror Part

**Mirror Part** creates a mirrored version of an existing part. This is a good way to create a left-hand version and a right-hand version of a part. Because the mirrored version is derived from the original version, the two parts always match.



This type of mirroring produces a different result than **Mirror Feature** (see page 4-44) or **Mirror All** (see page 4-45).

*To create a mirrored, derived part:*

- 1 In an open part document, click a model face or plane about which to mirror the part.
- 2 Click **Insert, Mirror Part**. The mirrored part is opened in a new part window.

## Derive Component Part

**Derive Component Part** creates a part from an assembly component. Derived parts created this way include any features created in the context of the assembly. For example, you can create a cavity in a mold assembly, then derive and cut the pieces of the mold. The derived parts maintain the necessary references to the components in the assembly.

See **External References** on page 7-2 for more information.

### ***To create a derived part from an assembly component:***

- 1 In an assembly document, select a component.
- 2 Click **File, Derive Component Part**. The derived part is opened in a new part window.

## Editing Derived Parts

Derived part documents have some limitations:

- You can only *add* features in a derived part document.
- You cannot delete or modify the features of the original part in the derived part document. You can only edit the original part in its own window.

### ***To edit the original part from which another part is derived:***

- 1 Right-click the derived part, and select **Edit In Context**.
  - If the part was derived using **Base Part** or **Mirror Part**, the original part document is opened.
  - If the part was derived from an assembly component, the assembly document is opened.
- 2 Change the original part as needed.

The changes are reflected in any derived parts that have an external reference to the original part.

## File Management with External References

When you save a part that has an external reference to another document, the path and the internal ID of the referenced document are saved with the current part. The software needs this information to locate and verify the original document the next time you open the derived part.

You can specify whether or not referenced documents are opened when you open a part that has external references. See **External References Options** on page A-11 (the **Load referenced documents** option).

Suppose you create a derived part, then you inadvertently move or rename the original document. When you try to open the derived part, the software notifies you that the externally referenced document (the original) was not found, and you are given a chance to look for it. If you choose not to look, the operation is cancelled.

If you decide to look for the referenced document, you have the following choices:

- If you locate and select the *original* document (in a different directory, or with a new name), the derived part opens using the new name or path. Because the internal ID matches, the external reference updates as expected. When you save the part, the new name or path is saved also.
- If you select a *different* document, you are notified that the internal ID does not match. Then you can either accept the selected document anyway, or keep looking.

If you accept the selected document, the model may have rebuild errors, especially if you have added features in the derived part document. If the geometry of the document you selected is not the same as the geometry of the original referenced document, the additional features may have rebuild errors.

## Checking Model Geometry

---

The SolidWorks software automatically checks the model geometry as you create each feature, to prevent invalid geometry. It also provides additional tools for checking the model, which you can use for the following purposes:

- To examine imported geometry (when you import the body as a whole, from another system, such as an IGES or STEP file)
- To make sure that the model exports properly to other formats
- To determine the cause when the model looks wrong

### Force Regeneration with Verification on Rebuild

By default, each time you add or modify a feature, the feature is checked against any adjacent faces and edges. For most situations, this default level of error checking is adequate, and results in fast rebuilding of the model.

However, under certain circumstances, it is recommended that you perform more rigorous checks on the model:

- When the model looks wrong (and you have determined that it is not the result of a display problem)
- When you have problem exporting the model in different formats

To increase the level of error checking, turn on the **Verification on rebuild** option. When this option is turned on, the software checks every new or changed feature against *all* existing faces and edges, not just adjacent faces and edges. Features that cause invalid geometry fail when this option is turned on.

---


**NOTE:** This option has a negative effect on performance. Rebuilding the model is considerably slower and more CPU-intensive. This option applies to *all* documents, not just the active document. Use this option only when needed, then be sure to turn it off when you are done.

---

#### **To verify the model with a higher level of error checking:**

- 1 Click **Tools, Options, Performance**, and select the **Verification on rebuild** check box.
- 2 Press **Ctrl+Q** to perform a *force regeneration* of the entire model.

---

**NOTE:** **Ctrl+Q** causes a complete rebuild of *all* the features in the model. The **Edit, Rebuild**  operation (**Ctrl+B**) only rebuilds *new or changed* features and their children.

---

- 3 Examine and correct any features that fail, beginning with the *first* failed feature in the FeatureManager design tree. Often fixing one failed feature allows subsequent failed features to rebuild successfully.
- 4 When you are satisfied that all the errors are corrected, click **Tools, Options, Performance**, and *clear* the **Verification on rebuild** check box.



## Check Entity

Even when **Verification on rebuild** is turned on, the model can still have errors. You can use **Check Entity** to identify any invalid or undesirable geometry.

### *To check a model:*

- 1 With a part document active, click **Tools, Check**.
- 2 In the **Check Entity** dialog box, select the type of check that you want to perform:
  - **Check body**. This is a fast check of the whole body that reports any errors on faces or edges. You can use this option first, to determine if the model geometry is valid. If you design very complex models, it is a good idea to use this option regularly, after creating several features.
  - **Check features**. If any errors are found when you use **Check body**, you can use this option to identify which features contain errors. This is slower, especially for complex models, because it examines the model feature by feature, to locate the source of the problem.
  - **Check selected item**. This checks a selected face or edge. The name of the selected item appears in the **Item to check** box.
- 3 Select the types of problems that you want to check for: **Invalid face(s)**, **Invalid edge(s)**, and **Short edge(s)**. When you check for **Short edge(s)**, you must specify a minimum edge length. The software reports any edges that are less than the specified length.
- 4 To find the smallest radius of curvature for the model, select the **Min. radius of curvature** check box. The face with the smallest radius of curvature is reported. If you chose the **Check selected item** option in Step 2, the smallest radius of curvature on the selected item is reported. You can select a model face, a spline, or a curve.
- 5 Click **Check**. (The dialog box remains open so that you can change the options or select different items to check.)
  - The **Found** column displays the number of errors of each type.
  - The **Result list** box lists the errors by name. To highlight the geometry, and to display a description of the error, select the item in the **Result list**.
- 6 Click **Close** to dismiss the dialog box.

## Custom Properties

---

You can use *custom properties* to add more information to a part or assembly document, or to a configuration of the model in the document. Custom properties can be used in several ways:

- In a bill of materials (see **Bill of Materials (BOM)** on page 10-31).
- As advanced selection criteria for managing assembly configurations (see **Component Selection by Properties** on page 6-47).
- As text in notes (see **Links to Document Properties** on page 10-21).
- In drawing templates (see **Customizing a Drawing Template** on page 9-5 and **Linking Notes to Document Properties** on page 9-7).

### ***To add custom properties:***

- 1 Click **File, Properties**.
- 2 To add properties *to the file*, click the **Custom** tab.  
To add properties *to the active configuration*, click the **Configuration specific** tab.
- 3 Enter a **Name** for the property, or choose a name from the list.

---

**NOTE:** If you plan to use the custom properties as columns in the bill of materials, do not use property names that contain spaces. See **Custom properties in a bill of materials** on page 10-36.

---

- 4 Select the **Type** of value (text, number, and so on) for the custom property.
- 5 Enter a **Value** for the custom property that is compatible with the selection in the **Type** box.
- 6 Click **Add**. The **Properties** box displays the name, value, and type of the custom properties.

### ***To modify custom properties:***

- 1 In the **Properties** box, click the **Name** of the property that you want to change.
- 2 Edit the **Type** or **Value** as needed.
- 3 Click **Modify**.

### ***To delete custom properties:***


In the **Properties** box, click the **Name** of the property that you want to delete, then click **Delete**.

## Displaying Annotations in Parts


---

You can add annotations to your model to further clarify the information the part document provides. You can add much of the necessary detailing of your models in the part document itself. This includes dimensions, notes, symbols, and so on. Then you can import the dimensions and annotations from the model into a drawing.


For more information about dimensions and annotations, see Chapter 10, “Detailing.”

You can control the display of annotations in the part window using shortcut menu selections on the **Annotations** folder  in the FeatureManager design tree.

### *To select the types of annotations to display:*

- 1 In the FeatureManager design tree, right-click **Annotations**  and select **Details**. The **Annotation Properties** dialog box appears.
- 2 Under **Display filter**, select the annotation types to display (**Notes**, **Datums**, and so on), or click the **Display all types** check box to show all types of annotations available for the part.
- 3 Change the values in the **Text scale** boxes to modify the scale of the text used in the annotation.
- 4 Select from the following options:
  - **Always display text at the same size.** When selected, annotations and dimensions are displayed at the same size regardless of zoom.
  - **Display items only in the view in which they are created.** When selected, an annotation is only displayed when the part is in the same view orientation as when the annotation was inserted. Rotating the part or selecting a different view orientation removes the annotation from the display.
  - **Display annotations.** When selected, all annotation types that are selected in the display filter are displayed.
- 5 Click **OK**.

### *To toggle the display of annotations:*

Right-click **Annotations**  and select (or clear) the items to display:

- **Display Annotations.** All annotation types that are selected in the **Annotation Properties** dialog box are displayed. This has the same effect as selecting **Display Annotations** in the **Annotation Properties** dialog box.
- **Show Feature Dimensions.** This has the same effect as selecting **Feature Dimensions** in the **Display filter** of the **Annotation Properties** dialog box.
- **Show Reference Dimensions.** This has the same effect as selecting **Reference Dimensions** in the **Display filter** of the **Annotation Properties** dialog box.

### ***To toggle the display of selected feature dimensions:***

To hide an individual dimension, right-click it, and select **Hide**.

To hide all the dimensions of a selected feature, right-click the feature in the FeatureManager design tree, or right-click one of its faces, and select **Hide All Dimensions**.

To re-display the dimensions, right-click the feature or a face, and select **Show All Dimensions**.

## **Section View of a Model**


---

You can display a view of the model cut through the part or assembly. You can use any combination of planes and planar model faces as section planes.

You may find it useful to view both the section view and the complete view of the model at the same time by splitting your screen into two windows. (Drag the split bars on the bottom or right side of the window frame.)

Before you can display a section view of an assembly, you must fully resolve any lightweight parts that it contains. See **Lightweight Parts** on page 6-40 and **Loading Additional Model Data** on page 6-41 for more information.

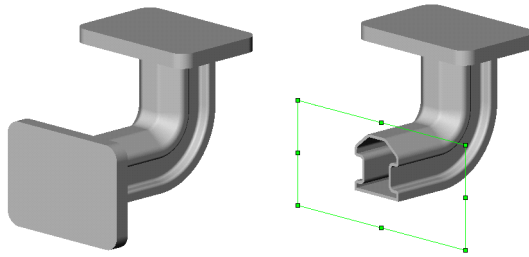
### ***To display a section view of the model:***

- 1 Click **View, Display, Section View**, or click **Section View**  on the View toolbar.
- 2 Click one or more planes in the FeatureManager design tree, or click one or more planar faces on the model.

Alternatively, you can pre-select planes or planar faces before clicking **View, Display, Section View**. If nothing is selected, **Plane1** is used by default. To pre-select multiple planes or faces, hold down the **Ctrl** key as you select.

The selected planes and faces are listed in the **Section Plane(s)/Face(s)** box.

- 3 Click a plane or face in the **Section Plane(s)/Face(s)** box, then enter an offset value in the **Section Position** box to move the section plane to the desired position for the cut.
- 4 Click the **Flip the Side to View** check box to reverse the viewing direction of the section cut.
- 5 Select a **Preview/Display** mode:
  - When **Preview** is selected, the section view is updated whenever any item in the dialog box is changed. If you need to adjust more than one setting, especially with a large part or assembly, this option may not be efficient.
  - When **Preview** is cleared, click **Display** to update the section view. You can make multiple changes to the settings, then click **Display** again to update the view with all the changes at once.
- 6 Click **OK**.



**To use the viewing plane to cut a section view:**

Instead of model faces and planes, you can use a *viewing plane* to cut the section view. The viewing plane is an invisible plane that is parallel to the screen, and normal to your line of sight. The initial position for this plane is the center of the part body bounding box. As you rotate the part, the angle of the viewing plane relative to the part body changes, cutting the part at a different angle.


To use the viewing plane to make a section cut, click **Use Viewing Plane** in the **Section View** dialog box.

You can offset the **Section Position** of the viewing plane. The **Flip the Side to View** option is not available when you use the viewing plane.

**To change the section view:**

- 1 Click **View, Modify, Section View**.
- 2 Change the settings in the **Edit Section View** dialog box, and select a different plane if needed.
- 3 Click **OK**.

**To return to a normal view of the model:**

Click **View, Display, Section View** (to deselect the command), or click **Section View**  again.

## Mass Properties and Section Properties


---

You can display the density, mass, volume, surface area, center of mass, inertia tensor, and principal axes of inertia of a part or assembly model.

Before you can calculate mass properties or section properties of an assembly, you must fully resolve any lightweight parts that it contains. See **Lightweight Parts** on page 6-40 and **Loading Additional Model Data** on page 6-41 for more information.

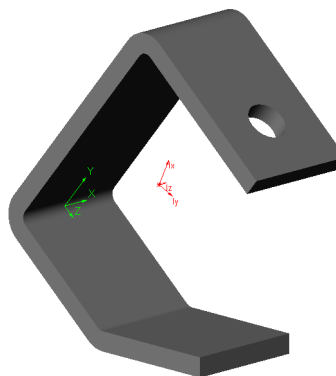
### Mass Properties

#### To display mass properties:

- 1 Click **Tools, Mass Properties**, or click **Mass Properties**  on the Tools toolbar. (To display the Tools toolbar, click **Tools, Customize**.)

To display the mass properties of an assembly component, select the component first. If no assembly component is selected, the mass properties for the entire assembly are calculated.

The results are displayed in the **Mass Properties** dialog box, and the principal axes and center of mass are displayed graphically on the model.



- 2 You can evaluate a different component without closing the **Mass Properties** dialog box. Select the component, then click **Recalculate**.
- 3 Click **Options** if you want to display the results using different units than the units specified for the active document. In the **Measurement Options** dialog box, select one of these systems of units:
  - **View measurement settings.** This is a *temporary* system of units. Changing the units here does not affect the units specified for the document, or the system default units (for new documents). You can specify the **Length unit**, **Angular unit**, and other options as needed (decimal places, display of fractions, and so on).
  - **View global settings.** Displays the units specified for the active document.
  - **View system defaults.** Displays the system default units for new documents.

You can view the settings for the active document and the system defaults, but you cannot change them here. (To change them, click **Tools, Options, Grid/Units**.)

To display the results using **Scientific notation**, select the check box.

Click **OK**, then click **Recalculate** to display the results in the selected units.

- 4 If you defined a coordinate system, you can display the results based on that system, instead of using the document's default coordinate system. Select the name from the **Output coordinate system** list, then click **Recalculate** to update the results.

See **Creating a Coordinate System** on page 3-7 for information about creating a user-defined coordinate system.

- 5 If you want to print the results directly from this dialog box, click **Print**.
- 6 If you want to copy the information to the clipboard, then paste it into another document, click **Copy**.
- 7 Click **Close** to close the dialog box.

### Section Properties

You can display certain mass properties for a planar face, sketch, or section face in a drawing view. The properties reported depend on the entity selected.

#### *To display section properties:*

- 1 Select a planar model face in any document, or the crosshatch section face in a section view in a drawing, or select a sketch (either click the sketch in the FeatureManager design tree, or right-click a feature, and select **Edit Sketch**).

---

**NOTE:** Section properties of sketches can only be calculated for *closed* profiles.

---

- 2 Click **Tools, Section Properties**. The results are displayed in the **Section Properties** dialog box, and the principal axes and center of mass are displayed graphically on the model.

The options and controls in the **Section Properties** dialog box are the same as those found in the **Mass Properties** dialog box. See the previous section for information about using them.

## Lighting

---


You can adjust the direction, intensity, and color of light in the shaded view of a model. You can add light sources of various types, and modify their characteristics to illuminate the model as needed.

The properties of the light sources work in conjunction with the **Material Properties** of the model (see page 5-3). Changing the material properties of the model can enhance or reduce the effect of the light properties.



### Light Types

There are several types of light source:

- Ambient** light illuminates the model evenly from all directions. In a room with white walls, the level of ambient light is high, because the light reflects off the walls and other objects.
- Directional** light comes from a source that is infinitely far away from the model. It is a collimated light source consisting of parallel rays arriving from a single direction, like the sun. The rays of a **Directional** light point toward the center of the model.
- Spot** light is a restricted, focused light with a cone-shaped beam that is brightest at its center. A spot light can be aimed at a specific area of the model. You can adjust the position and distance of the light source relative to the model, and the angle through which the beam spreads.
- Point** light comes from a very small light source located at a specific coordinate in the model space. This type of light source emits light in all directions. The effect is like a tiny light bulb floating in space.

By default, the **Lighting** folder  in the FeatureManager design tree contains the **Ambient** light. There is always exactly one **Ambient** light; you can turn it off, but you cannot delete it, or add another one. There is also one **Directional** light source by default.


#### *To add, delete, and turn off light sources:*

- To add a light, right-click the **Lighting** folder  or any existing light, and select **Add Spot Light (Directional Light, Point Light)**. The maximum number of lights in a document is nine (the **Ambient** light, plus eight others).  
You can also click **Insert, Light, Spot Light (Directional Light, Point Light)**.
- To delete a light, click the light icon in the **Lighting** folder  and press **Delete**.
- To turn off a light, without deleting it, edit the properties of the light (see the next section), and clear the **On** check box.



## Light Properties

### To modify the lighting properties:

- 1 Expand the **Lighting** folder  in the FeatureManager design tree.
- 2 Right-click the light to modify, and select **Properties**.
- 3 Edit the properties as needed. The light properties that are available depend on the type of light source. See the next section for more information.

As you edit each light, a graphical representation of the light source is displayed, indicating the color and direction of the light.

- 4 Click **OK**.

### Ambient Light Properties

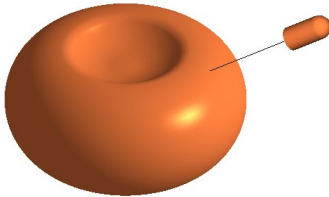
The properties on the **Basic** tab of the **Ambient** light source are common to all types of light sources.

- **Color.** To change the color, click **Edit**, select a new color from the palette, and click **OK**.
- **On.** To turn the light on or off, select or clear the **On** check box.
- **Use as default.** If you want to save the settings for the current light, and use them in new documents, click **Use as default**.
- **Intensity.** Move the **Ambient** slider to adjust the level of light. The level changes equally on all sides of the model.

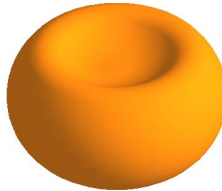
### Directional Light Properties

On the **Basic** tab, a **Directional** light source has the following additional **Intensity** properties:

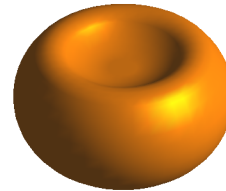
- **Brightness.** This slider controls the brightness of the light source. A higher value results in more light on the side of the model closest to the source.
- **Specularity.** This slider controls the extent to which shiny surfaces exhibit bright highlights where the light strikes them. A higher value results in more dramatic highlights and a shinier appearance.



Editing a **Directional** light



Higher values for  
**Ambient** and **Brightness**



Higher value for  
**Specularity**

On the **Direction** tab, specify the following properties:

- **Direction.** Specify the direction from which the light arrives, relative to the origin. You can use the sliders to specify the values for **Longitude and Latitude**, or you can enter a value in degrees.
- **Lock to model.** When this check box is *selected*, the light moves along with the model when you rotate the model. The light always shines on the same area of the model because the position of the light relative to the model is maintained.

To keep the light in the same position in the model space, clear this check box.

When you rotate the model, the light shines on different areas of the model because the position of the light remains fixed in space.

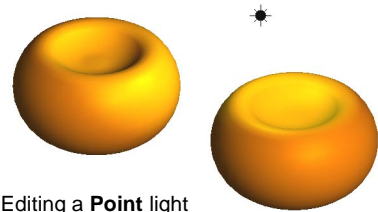
### **Point Light Properties**

On the **Basic** tab, specify the same properties for a **Point** light as you specify for a **Directional** light.

On the **Position** tab, specify the **Position** of the light source relative to the origin. Select the type of coordinate system, and specify the values:

- **Spherical (Longitude, Latitude, Distance)**
- **Cartesian (X, Y, Z)**

To lock the light in place, click **Lock to model**.

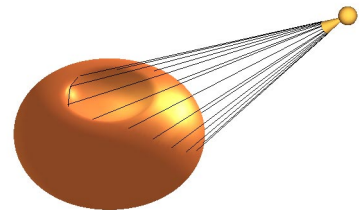


### **Spot Light Properties**

On the **Basic** tab, specify the same properties for a **Spot** light as you specify for a **Directional** light.

On the **Position** tab:

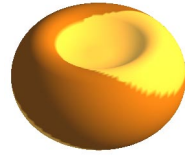
- **Position.** The **Position** is the location of the light source relative to the center of the scene. Like a **Directional** light, you can use either **Spherical** or **Cartesian** coordinates.
- **Target.** The **Target** is where the light strikes the model or the surrounding scene. Only Cartesian coordinates (**X, Y, Z**) are available.



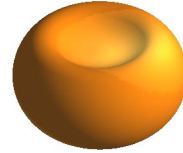
On the **Advanced** tab, you can specify the following properties:

- **Cone Angle.** Specify the angle through which the beam spreads. A small angle produces a narrow beam of light.
- **Exponent.** The exponent is the property that controls how tightly the beam of light is concentrated.

A low exponent value produces a focused, cone-shaped beam of light with sharp edges. The intensity of the light is similar at the center of the beam and at the edges.



With a high exponent value, the beam is brightest at the center. The intensity decreases toward the edges of the beam, and the edges of the beam appear softer.



- **Attenuation.** The attenuation is the property that decreases the intensity of the light as the distance increases. The values **A**, **B**, and **C** are multipliers in the following equation (**D** is the distance):

$$\text{attenuation} = \frac{1}{A + (B * D) + (C * D^2)}$$

As the values for **A**, **B**, and **C** increase, the amount of light that reaches the target decreases.

## Curvature

---

You can display a part or assembly with the surfaces rendered in different colors according to the local radius of curvature. Curvature is defined as the reciprocal of the radius ( $1/\text{radius}$ ), in current model units. By default, the greatest curvature value displayed is 1.0000, and the smallest value is 0.0010.

As the radius of curvature decreases, the curvature value increases, and the corresponding color changes from black (0.0010), through blue, green, and red (1.0000). You specify the color gradations associated with the curvature values on the **Color** tab of the **Options** dialog box.

As the radius of curvature increases, the curvature value decreases. A planar surface has a curvature value of zero ( $1/\infty$ ).

### ***To render curved surfaces in colors, and to display the radius of curvature values:***

- 1 Click **Tools, Options, Edges**, and make sure that the **Dynamic highlight from graphics view** check box is selected.
- 2 Click **View, Display, Curvature**. The curvature of the model is displayed in color. When you point to a model surface, a spline, or a curve, the curvature value and the radius of curvature are displayed beside the pointer.

To remove the color, click **View, Display, Curvature** again (to clear the check mark).

### ***To render an individual model face in colors:***

Right-click the model face and select **Face Curvature**. The curvature of the selected face is displayed. When you point to a model surface, the curvature value and the radius of curvature are shown beside the pointer.

To remove the color, right-click the model face and click **Face Curvature** again (to clear the check mark).

### ***To set values for the color scale for displaying curvature:***

- 1 Click **Tools, Options**, and select the **Color** tab.
- 2 Click the **Curvature** button.
- 3 Select the curvature value edit boxes that you want to change and enter new values. You can modify the values at five intervals on the color scale; values are interpolated for the intermediate colors in the range. The following table shows the curvature values for some common radius values.

Radius	Curvature	Radius	Curvature
0.125	8	10	0.1
0.25	4	20	0.05
0.5	2	25	0.04
0.75	1.333	30	0.033
1	1	40	0.025
1.5	0.667	50	0.02
2	0.5	75	0.013
2.5	0.4	100	0.01
3	0.333	125	0.008
4	0.25	200	0.005
5	0.2	250	0.004
7.5	0.133	500	0.002

- 4 Click the **Apply** button to see how the color scale changes in response to the new values.
- 5 Click **OK** to save the new settings.

## Sending Part Documents

---

You can send part documents by electronic mail. This function uses the electronic mail application installed on your computer.

***To mail a part document to another computer:***

- 1 With a part document open, click **File, Send To**.
- 2 Enter your mail password, the mailing address of the recipient, and any other information requested by your mail application.

The open document is attached to the mail message.

## Working with Assemblies

---

You can build complex assemblies consisting of many components.

This chapter describes the basics of creating an assembly from components you have already built, and general information about working with assemblies of any kind. The following topics are discussed:

- Design methods
- Creating an assembly
- Assembly mating
- Working with sub-assemblies
- Exploding an assembly view
- Simplifying large assemblies
- Using advanced component selection methods
- Customizing the appearance of an assembly
- Mailing an assembly file

## Design Methods

You can create an assembly using bottom-up design, top-down design, or a combination of both methods. This chapter explains the bottom-up method. For information about the top-down method, see Chapter 7, “Working with Parts within an Assembly.”

### Bottom-up Design

Bottom-up design is the traditional method. In bottom-up design, you create parts, insert them into an assembly, and mate them as required by your design. Bottom-up design is the preferred technique when you are using previously constructed, off-the-shelf parts.

An advantage of bottom-up design is that because components are designed independently, their relationships and regeneration behavior are simpler than in top-down design. Working bottom-up allows you to focus on the individual parts. It is a good method to use if you do not need to create references that control the size or shape of parts with respect to each other.

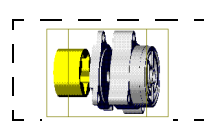
1. Create parts



2. Create assembly



3. Insert and mate parts

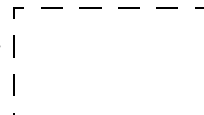


### Top-down Design

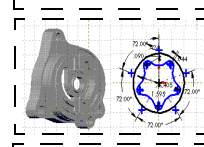
Top-down design is different because you start your work in the assembly. You can use the geometry of one part to help define other parts, or to create machined features that are added only after the parts are assembled. You can start with a layout sketch, define fixed part locations, planes, and so on, then design the parts referencing these definitions.

For example, you can insert a part in an assembly, then build a fixture based on this part. Working top-down, creating the fixture *in context*, allows you to reference model geometry, so you can control the dimensions of the fixture by creating geometric relations to the original part. That way, if you change a dimension of the part, the fixture automatically is updated.

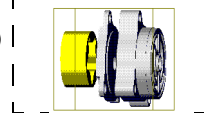
1. Create assembly



2. Create parts in place



3. Create (or insert) other parts



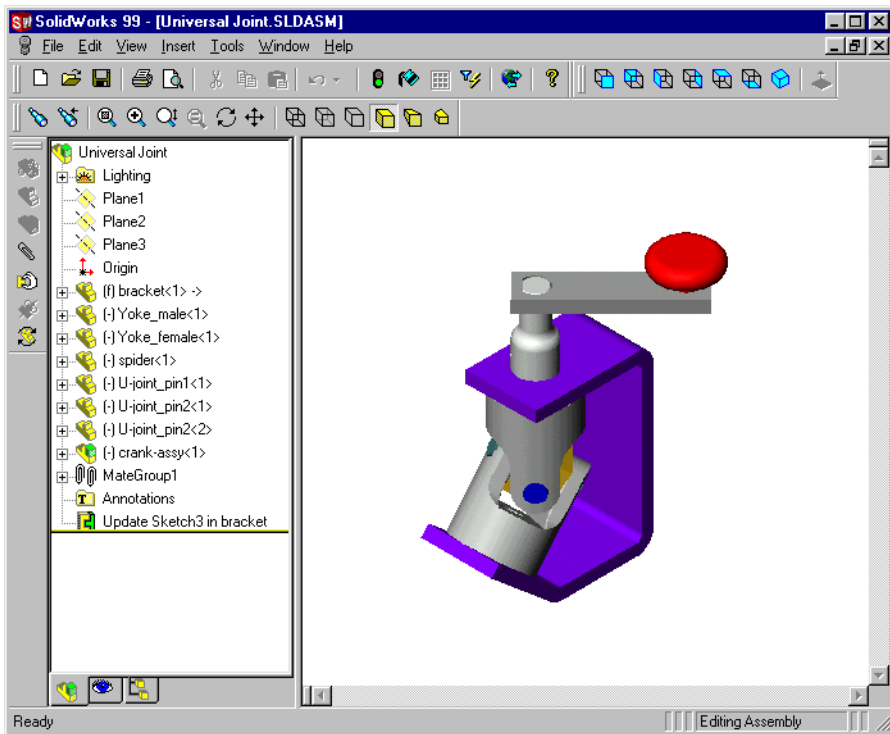
## Understanding Assemblies

The *components* of an assembly can include both individual parts and other assemblies, called *sub-assemblies*. For most operations, the behavior of components is the same for both types. See **Working with Sub-assemblies** on page 6-28 for complete information.

Components are *linked* to the assembly file. Assembly documents have the **.sldasm** extension.

### The Assembly Window


This is a typical assembly window.



The FeatureManager design tree displays the names of these items:

- Top-level assembly (the first item in the tree)
- Assembly planes and origin
- **Lighting** and **Annotations** folders
- Components (sub-assemblies and individual parts)
- Mategroups and mating relations
- Assembly features (cuts or holes) and component patterns
- Part features built in the context of the assembly



You can expand or collapse each component to view its detail by clicking the  beside the component name. To collapse all the items in the tree, right-click the assembly icon at the top of the tree, and select **Collapse Items**.

You can use the same component multiple times within an assembly. Each component has the suffix  $\langle n \rangle$ . For each occurrence of the component in the assembly, the number  $n$  is incremented.

In the FeatureManager design tree, a component name may have a prefix that provides information about the state of its relationships to other components. The prefixes are:

- (-) underdefined
- (+) overdefined
- (f) fixed
- (?) not solved

The absence of a prefix indicates that the component's position is fully defined.

### The Assembly Toolbar

The Assembly toolbar gives you quick access to these frequently used assembly tools.

-  **Move Component**
-  **Rotate Component Around Centerpoint**
-  **Rotate Component Around Axis**
-  **Mate**
-  **Edit Part**
-  **Hide/Show Component**
-  **Change Suppression State**

For information about moving and rotating components in an assembly, see **Positioning Components in an Assembly** on page 6-17.

For information about mating components, see **Assembly Mating** on page 6-19.

For information about editing a part in an assembly, see **Editing Parts in the Assembly Context** on page 7-6.

For information about hiding and showing components, or changing the suppression state, see **Simplifying Large Assemblies** on page 6-39.

## The FeatureManager Design Tree in an Assembly

There are times when you want to focus on the structure or hierarchy of the design rather than the details of the sketches and features. You can view an assembly's hierarchy in the FeatureManager design tree.

You may want to focus on the design of the assembly without all of the features of the components. By changing the FeatureManager design tree's display mode, you can view the assembly reference planes, components, assembly features, and mating relationships. The image of the assembly, including all its components (other than any suppressed or hidden ones), still appears in the graphics area.

Each of these ways of viewing the assembly affects only the level of detail displayed in the FeatureManager design tree. The assembly itself is not affected.

You can also use the FeatureManager design tree to modify the structure of the assembly. See **Working with Sub-assemblies** on page 6-28.

### ***To display the hierarchy or feature detail of an assembly:***

- To display the hierarchy, right-click the assembly name in the FeatureManager design tree, and select **Show Hierarchy Only**.

The FeatureManager design tree displays only the components (both parts and sub-assemblies), but no lower level detail.

- To display the detail again, repeat the procedure, selecting **Show Feature Detail**. To display the detail for a single component, double-click the component in the FeatureManager design tree.

### ***To view an assembly by dependencies or by features:***

- To display the dependencies, right-click the assembly name in the FeatureManager design tree, and select **View Dependencies**, or click **View, FeatureManager Tree, By Dependencies**.

Dependent items (instead of features) are listed under each component. Dependent items include mates and component patterns. This is a good way to identify all of the mates in which a component is used, and the patterns in which the component is a member.

- To display the features again, right-click the assembly name in the FeatureManager design tree, and select **View Features**, or click **View, FeatureManager Tree, By Features**.

## Creating an Assembly

---

An assembly can consist of parts you have previously designed as well as components that you create within the assembly. This section describes how to add, replace, copy, delete, verify, and save components in an assembly.

### Adding Components to an Assembly

When you place a component (either an individual part or a sub-assembly) in an assembly, the component file is linked to the assembly file. The component appears in the assembly; however, the data for the component remains in the source component file. Any changes you make to the component file update the assembly.

There are many ways to add components to a new or existing assembly.

- Use the command on the **Insert** menu, then browse to locate the component.
- Drag and drop from an open document window.
- Drag and drop from Windows Explorer.
- Drag and drop a hyperlink from Internet Explorer.
- Drag and drop within the assembly for additional instances of existing components.
- Drag and drop from the Feature Palette window (see **The Feature Palette Window** on page 11-6).

#### *To add components by using the menu:*

- 1 With an assembly open, click **Insert, Component, From File**. The **Insert Component** dialog box appears.
- 2 Browse to the folder that contains the component document that you want to insert into the assembly, and select the document. (You can also double-click the document to select it and close the dialog box in one step.)

To see the selected component before you insert it, click **Preview**.

To select which configuration of the component to use, click **Configure**. For information, see **Opening a Document in a Named Configuration** on page 8-5.

- 3 Click **Open**. If you clicked **Configure**, select a configuration from the list and click **OK**.
- 4 Click in the graphics area of the assembly window where you want to place the component.


#### *To add components by dragging and dropping from an open document window:*

- 1 Open the destination assembly document, and open the source document (either the component document itself, or another assembly document that contains the component).
- 2 Click **Window, Tile Horizontally** (or **Tile Vertically**).

- 3 Drag the component icon *from the FeatureManager design tree* of the source window, and drop it in the destination assembly window.

If you are adding an individual part (not a sub-assembly), you can also drag the part *from the graphics area*. Depending on the entity that you use to drag the part, you can create some mating relationships automatically when you drop the part. See **Smart Mates** on page 6-22.

If the component has multiple configurations, you can select which configuration to insert.

- To insert the configuration that is currently active, drag the icon from the top of the FeatureManager design tree of the component document, and drop it in the assembly window.
- To insert a configuration that is not currently active, click the Configuration tab  in the component document, then drag and drop the name of the desired configuration.

#### ***To add components by dragging and dropping from Windows Explorer:***

- 1 Open an assembly.
- 2 Open **Windows Explorer**. Browse to the folder that contains the desired component.
- 3 Drag the document icon from the **Explorer** window, and drop it in the graphics area of the assembly window.

If the component has multiple configurations, the **Select a configuration** dialog box appears. Select the configuration you want to insert, and click **OK**.

If the component has a **Mate Reference**, you can add certain types of mates automatically. See **Defining a Mate Reference** on page 6-25.

#### ***To add a component from a hyperlink in Internet Explorer:***

- 1 In Internet Explorer (version 4.0 or later), navigate to a location that contains hyperlinks to SolidWorks part files. You can only add individual parts, not sub-assemblies, from hyperlinks.

For example, on the SolidWorks website, in the Support section, navigate to the Solid Modeling Library page. You may also be able to obtain SolidWorks parts from some third-party vendors.

- 2 Drag a hyperlink from the Internet Explorer window, and drop it in the graphics area of the assembly window. The **Save As** dialog box appears.
- 3 Browse to the folder where you want to save the part, enter a new name if desired, and click **Save**.

The part document is saved to the specified folder. If the part has multiple configurations, the **Select a configuration** dialog box appears. Select the configuration you want to insert, and click **OK**. The part is added to the assembly.

**To add another component instance by dragging and dropping:**

With an assembly open, hold the **Ctrl** key, then drag a component either from the FeatureManager design tree, or from within the graphics area, and drop it in the graphics area.

When you add a new instance of a component, the instance number suffix (**component<n>**) is incremented.

**Adding Patterns of Components**

You can add a pattern of components in an assembly based on a feature pattern of an existing component. You can also define a pattern for placing components in an assembly in much the same way as you define a feature pattern in a part. Component patterns are *time-dependent*; they depend on an existing mating relation. See **Time-Dependent Features** on page 7-9 and **Adding a Component Pattern** on page 7-11 for more information.

**Inferencing to the Assembly Origin**

When you insert a component by dragging its name from the FeatureManager design tree of an open component window, or by clicking **Insert, Component, From File**, you can inference the assembly origin. Look for the pointer shown here when you point to the origin.



When you inference the assembly origin:

- the component is fixed (**f**)
- the component origin is coincident with the assembly origin
- the planes of the component and the assembly are aligned

You can also drag a part name from the FeatureManager design tree of a component window and drop it in the tree of the assembly window to inference the origin. The component is fixed only if it is the *first* component in the assembly.

**Deleting a Component from an Assembly**

**To delete a component from an assembly:**

- 1 Click the component in the graphics area or in the FeatureManager design tree.
- 2 Press the **Delete** key, or click **Edit, Delete**.
- 3 Click **Yes** to confirm the deletion. The component and all its dependent items (mates, component patterns, explode steps, etc.) are removed.

## Opening Assemblies

---

When you open an assembly document, the software accesses the information for all the components that are included in the assembly.

If any of the components has changed since the last time the assembly was opened, it is a good practice to rebuild the assembly before you continue working on it. You can specify whether you want assemblies to be rebuilt when you open them.

Click **Tools, Options, Performance**, and select one of these options under **Rebuild assembly on load**:

- **Never.** Open without rebuilding. This is a fast way to open an assembly when you only want to look at it, but you do not need to edit or save it.
- **Always.** Rebuild any assembly when it is opened, if any components have changed.
- **Prompt.** Ask each time an assembly is opened, if any components have changed. With this option, you can decide when you open each assembly whether or not you want to rebuild. When you open an assembly, click **Yes** or **No**.

If you also click **Don't ask me again**, the option selected under **Rebuild assembly on load** is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

By default, when you open an assembly, it is opened in the configuration that was in use the last time it was saved. You can also choose a different named configuration when you open the assembly. See **Opening a Document in a Named Configuration** on page 8-5.

---

**NOTE:** Whenever you open an assembly, if a referenced part or sub-assembly file cannot be found, you are asked to browse for it. You can also substitute a different part file, or the same part file with a new name. See **Replacing a Component in an Assembly** on page 6-12.

---

## Managing the Part Files of an Assembly


---

When you save an assembly, the path and name of each referenced component is saved in the assembly document. For this reason, you must exercise caution when you save or rename a component document, or move it to a different folder.

The sections that follow describe ways to keep the references in your assemblies up-to-date, and how to replace components with incorrect or missing references.

### Saving an Assembly and Its Parts

Under the **File** menu, there are several ways to save an assembly document. Whichever method you use, the current locations of all referenced components (parts and sub-assemblies) are saved with the assembly document.

- Save.** When you **Save** an assembly, you save the assembly document, and any referenced component documents that have changed.
- Save As.** When you use **Save As**, you save a copy of the assembly document with a new name or path. You can create a new folder  in the **Save As** dialog box if needed. After you use **Save As**, you are working in the new assembly document. The original assembly document is closed without saving.
- Save as Copy.** Click the **Save As Copy** check box in the **Save As** dialog box to save a copy of the assembly document, with a new name or path, *without* replacing the active document. You continue to work in the original assembly document.
- With both **Save As** and **Save as Copy**, if any of the components have changed since the last **Save** operation, you are notified that they need to be saved also. If you agree to save them, they are saved in their original referenced locations. See **Editing Referenced File Locations** on page 6-14 for information about saving referenced files in new locations.

### Verifying Which Components Are Used in an Assembly

You can check which part and sub-assembly files an assembly uses. This is useful if you have several versions of the component files.

To list the component files that are used in an assembly, click **File, Find References**. The **Search Results** dialog box displays a list of the components used in the assembly, including the full path names.

## Backing Up an Assembly

You can copy all component files used in an assembly to a new folder. This is an easy way to back up the assembly and all the referenced files.

### *To back up an assembly:*

- 1 Click **File, Find References**.
- 2 In the **Search Results** dialog box, click **Copy Files**.
- 3 In the **Choose Directory** dialog box, browse to the destination folder.
- 4 If you want the files to be stored with same structure (in subdirectories under the selected folder), click the **Preserve directory structure** check box.
- 5 Click **OK**. The assembly file and all the referenced files are copied to the new folder. This operation does not change the open assembly or any of its components; it simply makes a copy of all the files to the selected folder. The active assembly continues to reference the files in their original locations.

The following table lists the effects of saving an assembly in various ways. For this example, assume that these are the active documents:

assembly document	<b>c:\test\Assem1</b>
referenced part documents	<b>c:\test\PartA</b> <b>c:\test\PartB</b> (changed since the last save)

	<b>Saved files</b>	<b>Active assembly after saving</b>	<b>Referenced File Locations</b>
<b>Save</b>	c:\test\Assem1 c:\test\PartB	c:\test\Assem1	c:\test\PartA c:\test\PartB
<b>Save As</b> <b>c:\final\Assem2</b>	c:\final\Assem2 c:\test\PartB	c:\final\Assem2	c:\test\PartA c:\test\PartB
<b>Save As Copy</b> <b>c:\final\Assem2</b>	c:\final\Assem2 c:\test\PartB	c:\test\Assem1	c:\test\PartA c:\test\PartB
<b>Find References,</b> <b>then Copy Files to</b> <b>c:\final</b>	c:\final\Assem1 c:\final\PartA c:\final\PartB	c:\test\Assem1	c:\test\PartA c:\test\PartB



## Replacing a Component in an Assembly

An assembly and its components may go through many revisions during the course of a design cycle. Managing the referenced files becomes very important when the referenced documents change.

This is especially true in a multi-user environment. Several users can work on the individual parts and sub-assemblies. Then all the components are brought together and managed in the top-level assembly. As components are added and modified, the references in the top-level assembly need to be updated so that the latest revisions are used.

You could delete the out-of-date components, then insert the new versions. However, deleting a component also deletes its mates, and breaks its external references for in-context features. To restore the assembly to its previous condition, you have to add the mates again, and fix any failed in-context features.

A safer, more efficient way to update the assembly is to *replace* the components as needed. Depending on the differences between the original component and the replacement component, the mates and in-context features may not be affected at all.

There are several ways to replace assembly components.

- ❑ **Use Reload/Replace.** This is an easy way to replace all instances of a single component, either with an updated version of the same file, or with a new file. This is especially useful when several people work on the individual components. You can reload or replace the components to update the assembly with current versions.
- ❑ **Edit the component properties.** In addition to replacing a component, editing the component properties allows you to specify a different name for identifying the component in the FeatureManager design tree. This affects only the selected instance.
- ❑ **Open the assembly with new referenced file locations.** This is an efficient way to replace several components at once, especially in these situations:
  - When multiple replacement components are stored together in a different folder
  - When the files have a common text string in their names that you can find and replace (revision names, for example)
- ❑ **Save the assembly with new referenced file locations.** Use this method to save the assembly and its components in a new location, and update the referenced file locations at the same time.

This is different from **Save as Copy** (see page 6-10). After this operation, the assembly references the files in the new locations you specify (not the files it referenced when you opened it). This is especially useful when you create in-context features; you can save and reference a copy of the part that includes the in-context features, and leave the original part file unchanged for future use.

**To reload or replace an assembly component:**

- 1 In the FeatureManager design tree or the graphics area, right-click the component you want to reload or replace, and select **Reload**.
- 2 Click the **Open as read-only** check box if you only need read access, while another user needs write access to the document.
- 3 Select either **Reload** or **Replace**.
  - **Reload** refreshes shared documents. If you open a component document for read-only access and use that component while another user is making changes to it, you can reload it to get the updated version.
  - **Replace** lets you replace all instances of the selected component with a new component instead. Click **Browse** to find the replacement component.
- 4 Click **OK** to accept the change, or click **Cancel**.

**To replace a single instance of a component by editing its properties:**

- 1 In the FeatureManager design tree or the graphics area, right-click the component you want to replace, and select **Component Properties**.
- 2 Enter a **Component Name** if you want the FeatureManager design tree to list the component by a name that is different from the name of the component file.

If the **Component Name** box is unavailable, click **Tools, Options, External References**, and clear the **Update component names when documents are replaced** check box.

- When selected (the default), the component name in the FeatureManager design tree is the name of the component file. If you replace a component, the name in the tree updates to use the name of the replacement component file.
- When cleared, you can specify the component name to be used in the FeatureManager design tree. If you replace a component, the name in the tree does not change.

If your naming convention includes information about the revision, date, etc., this allows you to use a generic name (**housing**), instead of the component file name (**housing\_revA3**).

The **Component Name** is used *in the FeatureManager design tree only*; the component's file name does not change. Component names may not include the forward slash (/) or at (@) characters.

- 3 To locate the replacement component, click **Browse**. Navigate to the new document. If desired, click the **Preview** check box to view the component in the **Preview** window.
- 4 Click **Open**. The **Model Document Path** displays the selected document name.
- 5 Click **OK** to replace the component.

The last saved or in-use configuration of the selected component is used. For more information about configurations, see **Component Configurations in an Assembly** on page 8-7.

## Editing Referenced File Locations

When you need to replace all the instances of several components, editing the referenced file locations is an efficient method. You can change the references so that the assembly document looks in a different folder to locate the component documents. You can also find and replace text strings contained in the document names (for example, a suffix that indicates the revision number).

For the following examples, assume that the assembly document currently references these parts:

**c:\in\_progress\partA\_rev1**

**c:\in\_progress\partB\_rev1**

You want to update the assembly to reference these parts:

**c:\completed\partA\_rev2**

**c:\completed\partB\_rev2**

### ***To open an assembly with new references:***

This operation changes the referenced file locations *as you open* the assembly. Instead of using the referenced file locations that were saved with the assembly, you specify new references now.

- 1 Click **File, Open**, select the assembly, and click the **References** button.

The **Edit Referenced File Locations** dialog box lists the **Current pathname** (and the same **New pathname**) for the components.

- 2 Click the referenced documents that you want to change, or click **Select All**.
- 3 To change the folder where the documents are stored, click **Browse**, then locate the new folder (**completed**, in this example). When you click **OK** in the **Choose Directory** dialog box, the **New pathname** is updated for the selected documents.
- 4 To change the names, click **Replace**. In the **Find what** and **Replace with** boxes, enter the text strings to find and replace (**rev1** and **rev2**, in this example).

To find occurrences of the string one by one, click **Find Next**, then **Replace** where needed. Otherwise, click **Replace All**.

- 5 Click **OK**. If any of the documents does not exist in the new folder, a red exclamation mark indicates which ones were not found, and you can try again.
- 6 Click **Open**. The assembly opens, using the updated reference locations.

---

**NOTE:** You do not have to update the path and the name at the same time.

---

- 7 Save the assembly. This is necessary to save the new reference locations in the assembly document.

### **To save an assembly with new references:**

This method saves the assembly and its components in a new location, and updates the referenced file locations at the same time. After this operation, you are working in the newly saved assembly document, with the new references. The original assembly document is closed without saving.

- 1 Click **File, Save As**, specify the folder and filename for saving the assembly document, then click **References**.
- 2 Select the referenced documents that you want to save with new names or paths. To change the path for all the items in the list, click **Select All**.
- 3 To edit the path of the selected documents, click **Browse**, then navigate to the target folder (**completed**, in the example).
- 4 To edit the names of the selected documents, click **Replace**, enter the text strings to find and replace (**rev1** to **rev2** in this example), and replace as needed.

---

**NOTE:** You can also click-pause-click the name in the **New pathname** list, then type the path to an existing folder, and the new name.

---

- 5 Click **OK**, and click **Save**. If the specified documents already exist, you can choose to replace them; if they do not exist, the assembly and the components are saved with the new paths and names.

You are now working in the newly saved assembly, which references the newly specified file locations.

When an assembly has in-context features, there are some special considerations for saving referenced file locations. See **Saving Assemblies with In-context Features** on page 7-5.

## Preserving Mates with Component Replacement

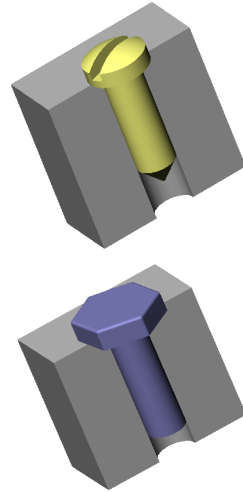
If the original part and the replacement part are similar, mates used in the original part are applied to the replacement part wherever possible. Preservation of the mates depends on the entities used in the mates being the same. (See **Assembly Mating** on page 6-19 for more information about mates.)

If you replace one part with another part created by deriving or copying, the entities used in the mates are likely to match (depending on the differences between the parts), and the mates will solve as expected.

For example, this assembly maintains the concentric mate between the cylindrical faces, and the coincident mate between the flat faces (the block face and the bottom face of the head on the screw or bolt).

If the entities used in the mates are not the same, or do not exist, in the replacement part, the mate will have an error. You can either delete the failed mate, or edit the mate to use a different entity.

You can rename the corresponding edges and/or faces on a replacement part to match the edge/face names on the original part to ensure that the mates are preserved.



### ***To automatically name the entities used in a mate:***

Click **Tools, Options, External References**, and click **Automatically generate names for referenced geometry**. You should select this option before you add mates if you intend to perform component replacement using the same surface identifiers.

- When selected, surface identifiers (**Face1, Edge2**) are created automatically in each part when you add a mate; therefore, you need write access to the parts, so that the identifiers can be saved in the part documents.
- When cleared (the default), you can mate to parts for which you have read-only access because you use the internal face IDs of the parts. If you do not plan to replace components, leave this option off, especially in a multi-user environment.

If you need to replace components in an assembly whose mates were added when the option was cleared, you can still rename the entities individually.

### ***To manually name the entities used in a mate:***

- 1 In the *original* part document, right-click the entity used in the mate, and select **Face Properties** (or **Edge Properties**).
- 2 Under **Entity Information**, enter a new **Name**, and click **OK**.

- 3 In the *replacement* part document, right-click the corresponding edge or face, and edit the entity properties, using the same **Name**. Now when you replace the component, the mate can be preserved.

If the replacement component is derived from the original one, you only need to enter names for the entities that differ between the two parts.

**To identify named entities:**

- 1 In the part document, right-click the part name at the top of the FeatureManager design tree, and select **List Named Entities**. The list includes names added both manually and automatically.  
If the part has no named entities, the shortcut menu does not include this selection.
- 2 To highlight a named entity, click the item in the **Named Entities** dialog box.

## Positioning Components in an Assembly

---

Once a component is placed in an assembly, you can move it, rotate it, or fix its location. This is useful for rough placement of the components in the assembly. You can then position the components precisely using mating relationships.

As you add mating relationships, you can move the components within the unconstrained degrees of freedom, visualizing the mechanism's behavior.

### Fixing the Position of a Component

You can fix the position of a component so that it cannot move with respect to the assembly origin. By default, the first part in an assembly is fixed; however, you can unfix it at any time.

It is recommended that at least one assembly component is either fixed, or mated to the assembly planes or origin. This gives a frame of reference for all other mates, and helps prevent unexpected movement of components when mates are added.

**To fix a component:**

In either the FeatureManager design tree or the graphics area, right-click the component, and select **Fix**.

In the FeatureManager design tree, the prefix **(f)** appears next to the name of the fixed component. A fixed component is locked in place and cannot move.

**To float (unfix) a component:**


In either the FeatureManager design tree or the graphics area, right-click the component, and select **Float**.

In the FeatureManager design tree, the prefix changes to reflect the status of the component before its position was fixed.

## Moving a Component


When you move a component, it retains any mating relationships (within its own mategroup) with other components. You cannot move a component whose position is fixed or fully defined. You can only move a component within the degrees of freedom allowed by its mating relationships.

### To move a component:

- 1 Click **Move Component**  on the Assembly toolbar, or **Tools, Component, Move**.
- 2 Drag a component to move it. You can move one component after another; the tool remains active until you click it again, or select another tool.

You can select a component to move in the FeatureManager design tree also. Dragging the mouse anywhere in the graphics area moves the selected component.

---


**NOTE:** Clicking **Pan**  on the View toolbar moves the view of the *entire assembly*.

---


## Rotating a Component

There are two ways to rotate a component: you can rotate it freely around its centerpoint, or you can rotate it around an axis. Only components that have not yet been mated can be freely rotated. A mated component can be rotated around an axis if its mates allow it. When rotating a component around an axis, all mating relationships within the component's own mategroup are maintained.

### To rotate a component freely:


- 1 Click **Rotate Component Around Centerpoint**  on the Assembly toolbar, or click **Tools, Component, Rotate**. You can only use this rotate tool on components that have not been mated.
- 2 Drag to rotate the component. You can rotate one component after another; the tool remains active until you click it again, or select another tool.

---

**NOTE:** Clicking **Rotate View**  on the View toolbar rotates the view of the *entire assembly*.

---

### To rotate a component around an axis:

- 1 Click an axis, linear edge, or sketch line around which to rotate the component.
- 2 Hold the **Ctrl** key and click the component to rotate.
- 3 Click **Rotate Component Around Axis**  on the Assembly toolbar.
- 4 Drag to rotate the component around the axis.

## Measuring Distances Between Components

You can measure the size of items, and the distance and angle between lines, points, surfaces and planes in sketches, part or assembly models, or drawings. See **Measuring Sizes and Distances** on page 5-11 for more information.

# Assembly Mating

## Mating Relationships

Mating relationships let you precisely position the components with respect to each other in an assembly. They let you define how the components move and rotate with respect to other components. By adding mating relationships successively, you can move the components into the desired positions.

Mating creates geometric relationships, such as coincident, perpendicular, tangent, and so on. Each mating relationship is valid for specific combinations of geometry. The following table shows the mating relationships that are supported between the various types of geometry.

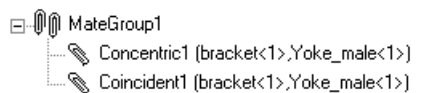
**Table 6-1. Geometry types supported in mating relationships**

	Plane	Cylinder	Line	Cone	Point
<b>Plane (planar face or plane)</b>	coincident distance parallel perpendicular angle				
<b>Cylinder (cylindrical face)</b>	tangent	tangent concentric			
<b>Line (linear edge, axis, or sketch line)</b>	coincident parallel perpendicular distance	coincident tangent concentric	coincident parallel perpendicular distance angle		
<b>Cone (conical face)</b>	---	concentric	concentric	concentric	
<b>Point (origin, vertex, or sketch point)</b>	coincident distance	coincident concentric	coincident distance	concentric	coincident distance

## Mategroups

When you create a new assembly, an empty mategroup (with the default name **MateGroup1**) is automatically included in the FeatureManager design tree. Each mating relationship that you create is added to a mategroup. Click the **+** to expand a mategroup and see the mates that it contains.

In the FeatureManager design tree, the entry for each mate includes the names of the components involved.





Assemblies can have multiple mategroups. Additional mategroups are automatically added when needed. This occurs when performing certain operations, such as adding a mate to a time-dependent feature in the assembly. See **Time-Dependent Features** on page 7-9 for more information.


All the mates within a mategroup are solved together; the order in which they appear within the mategroup does not matter.


## Creating a Mating Relationship

There are several ways to create mating relationships between components. This section describes how to use the options in the **Assembly Mating** dialog box.

See **Smart Mates** on page 6-22 for information about adding some types of mates automatically while dragging the components.

### *To mate components in an assembly:*

- 1 Click **Mate**  on the Assembly toolbar or click **Insert, Mate**.

In the **Assembly Mating** dialog box, you can click the pushpin  to keep the dialog box open. This allows you to apply multiple mates before closing the dialog box.



- 2 Select the desired faces, edges, vertices, or reference planes on the components. The entities for each component are listed in the **Items Selected** box.

To make selection of the desired entity type easier, use the **Selection Filter**.

---

**NOTE:** For components with drafted or curved faces, it is sometimes easiest to create mating relationships between the planes and sketches of the components and/or the assembly.

---

If you need to move or rotate the assembly to select entities for mating, click **Rotate View**  or **Pan**  on the View toolbar. Move the model as needed, then click the tool again. You can also use the arrow keys to change the model position. While you are adding a mate, you can only move the assembly as a whole; you cannot move individual components.

- 3 Select the type of mate you want under **Mate Types**. Only the mating types that are valid for the selected entities are available. If you select **Distance** or **Angle**, you must also specify the value.

---

**NOTE:** A **Distance** mate with a value of zero is more flexible than a **Coincident** mate. In some cases (if the assembly does not explode as expected, or if other mates do not solve), this may be a better choice.

You can link the value of a **Distance** or **Angle** mate to a sketch dimension.

---

4 Select an **Alignment Condition**.

- **Aligned.** The normal vectors for the selected faces point in the *same* direction.
- **Anti-aligned.** The normal vectors for the selected faces point in *opposite* directions.
- **Closest.** The components are either aligned or anti-aligned, depending on which condition can be satisfied with the least movement.

5 In the **Workbench** area of the dialog box, click **Preview** to see what the assembly will look like when the mate is performed.

6 If you need to reverse the direction of the measurement for a **Distance** or **Angle** mate, click the **Flip Dimension to Other Side** check box. Click **Preview** again to see the effect.

The examples in Table 6-2 show the effect of the changing the alignment condition for **Coincident** and **Distance** mates, and flipping the direction for a distance mate.

All the examples use the same face selections. Section views of the model are shown for clarity.

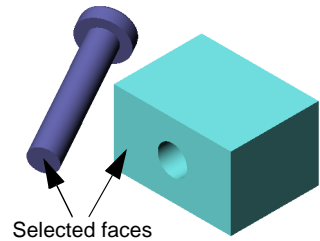


Table 6-2. Alignment conditions in mating relationships

	Aligned	Anti-aligned
<b>Coincident</b>		
<b>Distance</b>		
<b>Distance, Flip Dimension to Other Side</b>		

- 7 If the mate does not look the way you want it to look, click **Undo**, change the options as needed, and click **Preview** again.
- 8 If you want to define several mates, then solve them all at once, click the **Defer Mate** check box. Mates defined while this check box is selected are solved when you either close the dialog box or clear the check box.
- 9 Click **Apply** when you are satisfied with the mate. When you close the **Assembly Mating** dialog box, any deferred mates are solved, and the components move into the specified positions.

## Modifying a Mating Relationship


### *To modify a mating relationship:*

- 1 Expand the **MateGroup** in the FeatureManager design tree if necessary. Right-click the mate, and select **Edit Definition**.

The related geometry is highlighted in the graphics area.

- 2 In the **Assembly Mating** dialog box, change the desired options.
  - **Alignment Condition (Aligned or Anti-aligned)** on mates which imply a direction
  - **Flip Dimension To Other Side** to reverse the direction in which a **Distance** or **Angle** mate is measured
  - Value of an **Angle** or **Distance** mate
  - You *cannot* change the mate type.
- 3 To replace one entity with another, click the entity to replace in the **Items Selected** box, press the **Delete** key, then select the new entity.
- 4 Click **Apply** to make the change.

---

**NOTE:** To change only the dimension value of a **Distance** or **Angle** mate, double-click the mate in the FeatureManager design tree, then double-click the dimension to edit. Enter the new value, then click **Rebuild** .

---

## Smart Mates

You can create some types of mating relationships automatically. Mates created with these methods are referred to as *smart mates*.

- You can create mates *when you insert* a part in the assembly, by dragging the part in specific ways from an open part window. The entity that you use to drag determines the types of mates that are added.
- You can specify a *mate reference* in the part document. A mate reference identifies the entity to use for mating when you insert the part from a file. See **Defining a Mate Reference** on page 6-25.
- You can create mates by *inferring* potential mate partners *when you move* a part within the assembly. This is available for any under defined part, even if it is used in other mates. See **Inferring Mates While Moving Components** on page 6-26.

## Geometry-Based Mates

You can create some types of smart mates by dragging parts into assemblies using certain model geometry.






### To create smart mates while adding a component to the assembly:

- 1 Drag a part from an open part window into the graphics area of the assembly window, *but do not drop it immediately*. Use one of the following entities to drag the part: a linear or circular edge, a temporary axis, a vertex, a planar face, or a cylindrical/conical face.

When the pointer is over an entity of another component that is a potential mate partner, the pointer changes to indicate the mate or mates that will result if the part is dropped at this location. A preview of the part snaps into place.

- 2 If the preview indicates that you need to change the **Alignment Condition**, press the **Tab** key to flip the alignment (**Aligned/Anti-aligned**).
- 3 Drop the part. The mates are added automatically. The following table describes the types of automatic mates that are supported.

**Table 6-3. Smart mates**

Mating entities	Type of mate	Pointer
linear edge to linear edge	coincident	
planar face to planar face	coincident	
vertex to vertex	coincident	
axis to axis conical face to axis axis to conical face* conical face to conical face*	concentric	
circular edge to circular edge	coincident (planar faces) and concentric (conical faces*)	

\* Wherever *conical* faces are specified, *cylindrical* faces can also be used. The faces must be of the same type: either two conical faces or two cylindrical faces, not one of each. For circular edge mating, there must also be a planar face adjacent to each conical face.

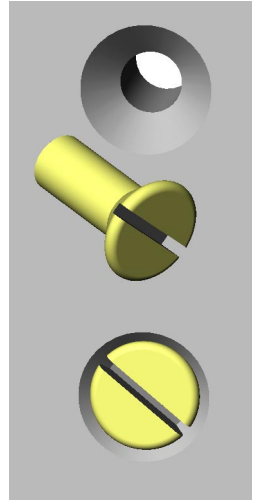
## Feature-Based Mates

You can also add smart mates between conical or cylindrical features that have a “peg-in-hole” relationship. One must be a base or boss, and the other must be a hole or a cut. The features may be extruded or revolved, and a cross-section of the feature must be a complete circle.

The features must have a matching set of faces. The faces must be of the same type (both conical or both cylindrical, you may not use one of each type). There must also be a planar face adjacent to the conical/cylindrical face.

A **Concentric** mate is added between the conical/cylindrical faces. The way the part is designed, and the feature that you select, determine the entities used for the **Coincident** mate between the planar faces. For example:

- If you mate the cylindrical base of a screw to the cylindrical face of a countersunk hole, the planar face at the end of the screw mates to the planar face at the bottom of the hole.
- If you mate the conical boss of the screw to the conical face of the hole, the planar face on the head of the screw mates to the planar face at the top of the hole.
- If you create the whole screw as a single revolved base, then mate it to either face of the hole, the top planar faces are mated. When the set of faces matches this way, the coincident mate is applied where the faces meet at the largest circular edge.



### To create smart mates using features:

- 1 In the FeatureManager design tree of a part document, select a feature with a cylindrical or conical face. Drag the feature name into an assembly graphics window.

When the pointer is over another cylindrical or conical face, it changes to indicate the mates that will result if the part is dropped at this location.



A preview of the part snaps into place. If the preview indicates that you need to change the **Alignment Condition**, press the **Tab** key to flip the alignment (**Aligned/Anti-aligned**).

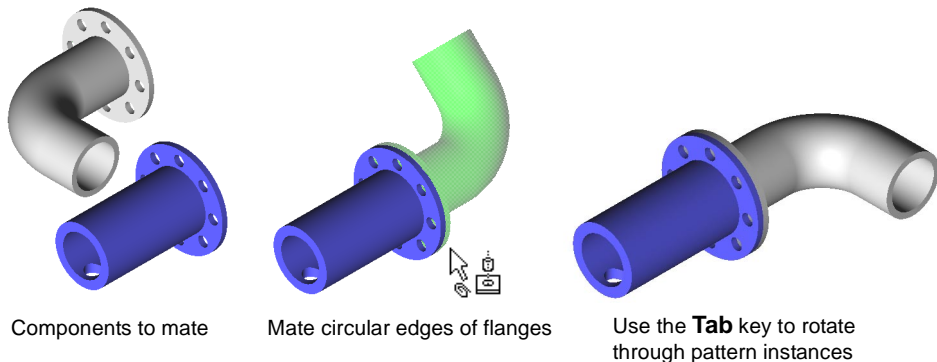
- 2 Drop the part. Two mates are applied: a **Concentric** mate between the cylindrical or conical faces, and a **Coincident** mate between the planar faces that are adjacent to the conical faces.

## Pattern-Based Mates

In some special cases, you can create as many as three smart mates at once. Each part must have a circular pattern of cylindrical holes (or bosses) on a planar face with a circular edge. Drag the part into the assembly using the circular edge.

A **Concentric** mate and a **Coincident** mate are added, consistent with the geometry-based mates described earlier. Then, if possible, a third **Concentric** mate is added, between a pattern instance on the part that you are dragging and one on the part that is already in the assembly.

Press the **Tab** key to rotate the part that you are dragging, to align it with the desired pattern instance.



## Defining a Mate Reference

A mate reference identifies which entity of a part is used for smart mates when you insert a component. The software can identify potential mate partners for the specified entity. This lets you take advantage of smart mates when you add parts by dragging them from the Feature Palette window or from Windows Explorer. You do not have to open the part in its own window, in order to drag by a suitable mating entity.

The mate reference is also used if you drag a part icon from the top of the FeatureManager design tree into the assembly window.

As you drag the part into the assembly window, the pointer changes and the preview snaps into place when a potential mate partner is found.

### **To define a mate reference:**

- 1 In a part document, click **Tools, Mate Reference**.
- 2 Select an entity to use for automatic mating, and click **OK**. You can use a linear or circular edge, a vertex, or planar or conical face. A part can have only one mate reference.

To select a different entity, right-click the **MateReference** feature in the FeatureManager design tree, and select **Edit Definition**. Select a new entity, and click **OK**.

## Inferring Mates While Moving Components



You can add smart mates when you move a component in an assembly, even if the component is used in other mates. As you drag the component into place, you can infer potential mate partners for creating geometry-based smart mates (see the table on page 6-23).

---

**NOTE:** If a component is already used in other mates, you can only move it within the degrees of freedom allowed by those mates.

---

### *To create smart mates while moving a part:*

- 1 Select a mating entity (face, edge or vertex) on a component that you want to move and mate.
- 2 Click **Move Component**  on the Assembly toolbar.
- 3 Drag the selected component onto another component until the smart mate pointer indicates the mates that will be added. To flip the alignment condition, press **Tab**.  
– or –  
Click the corresponding mating entity on the second component.
- 4 To add more mates while the **Move Component**  tool is active, double-click the mating entity on another component, and repeat Step 3.

When you move a component with a mating entity selected, the component is displayed with a transparent appearance. This indicates that you can infer potential partners for smart mates while dragging.

To display the component with its normal (not transparent) appearance, hold the **Shift** key as you drag. This can help you see what the assembly will look like when the mate is completed. To return to the transparent appearance, release the **Shift** key.

To temporarily suspend the inferring of smart mates while dragging, press the **Alt** key. You can then drag the component in the normal way. When you want to infer potential mate partners again, press **Alt** again.

## Deleting a Mating Relationship

You can delete mating relationships when necessary. When you delete a mate, it is deleted in *all* configurations of the assembly.

### *To delete a mating relationship:*

- 1 Click the desired mating relationship in the FeatureManager design tree.
- 2 Press the **Delete** key, or click **Edit, Delete**.
- 3 Click **Yes** to confirm the deletion.

## Suppressing a Mating Relationship

You can suppress mating relationships to prevent them from being solved. This allows you to experiment with different types of mates without over defining the assembly. When you suppress a mate, it is suppressed only in the *active* configuration of the assembly. See **Creating a Configuration Manually** on page 8-2 for more information.

### *To suppress a mating relationship:*

- 1 Right-click the mating relationship in the FeatureManager design tree, and select **Properties**.
- 2 Click the **Suppressed** check box, and click **OK**.

To unsuppress the mate, repeat the process, and clear the **Suppressed** check box.

You can also select one or more mates (hold the **Ctrl** key to select more than one), and click **Edit, Suppress** (or **Unsuppress**).

## Errors in Solving Mates

If changes in a component remove an entity (face, edge, etc.) that is involved in a mate, the mate cannot be solved. The mate is not deleted, however. The mate is suppressed, and is displayed in the FeatureManager design tree with an error symbol. Suppressed mates are not considered when the remaining mates are solved.

Sometimes adding many mates can cause the position of a component to be over defined. To correct the situation, suppress the mates on the component individually, then test the degrees of freedom by dragging the component. That way you can identify and delete the mate that is causing the conflict.

To make this task easier, list the items in the FeatureManager design tree by dependencies. See **The FeatureManager Design Tree in an Assembly** on page 6-5.




## Working with Sub-assemblies

---

When an assembly is a component of another assembly, it is referred to as a *sub-assembly*. You can nest sub-assemblies in multiple levels, to reflect the hierarchy of your design.

---

**NOTE:** Operations that change the structure of an assembly clear the **Undo** list in the active assembly. After performing such operations, you cannot click **Undo**  to restore the assembly to its previous condition.

---

### Creating a Sub-assembly

There are several ways to create a sub-assembly:

- You can create an assembly document as a separate operation, then make it a sub-assembly by inserting it into a higher-level assembly.
- You can insert a new, empty sub-assembly at any level of the assembly hierarchy while you are editing a top-level assembly, then add components to it in a variety of ways.
- You can form a new sub-assembly by selecting a group of components that are already in the assembly. This creates a sub-assembly and adds components to it in a single step.

No matter how you create the sub-assembly, you can open it for editing in its own assembly window, or you can edit it in the context of a higher-level assembly.

### Inserting a New Sub-assembly

You can insert a new, empty sub-assembly at any level of the assembly hierarchy. Then you can add components to it in a number of ways.

#### **To insert a new sub-assembly:**


- 1 In the FeatureManager design tree, right-click either the top-level assembly icon or the icon for an existing sub-assembly, and select **Insert New Sub-assembly**.

You can also select an assembly icon and click **Insert, Component, New Assembly**.

The **Save As** dialog box appears.

- 2 Browse to a different folder if needed, enter a **File name** for the new assembly document, and click **Save**.

An empty sub-assembly is inserted as the last component of the selected assembly, and a new assembly document (**.sldasm**) is created and saved in the folder you specify.

- 3 Add components to the sub-assembly:
  - To *move a component* that is already in the assembly into the sub-assembly, drag the component name in the FeatureManager design tree, and drop it on the sub-assembly icon. Look for the  pointer.

You can also click **Tools, Reorganize Components** to move components into the sub-assembly.

You can move components (individual parts or sub-assemblies) from any level into the new sub-assembly. See **Modifying a Sub-assembly** on page 6-30 for information about the effects of moving components.

- To *add an existing, saved component* to the sub-assembly, right-click the sub-assembly icon in the FeatureManager design tree, and select **Edit Sub-assembly**. Then insert components using any of the methods in **Adding Components to an Assembly** on page 6-6.
- To *insert a new component* into a sub-assembly, you must edit the sub-assembly in its own window. You cannot insert a *new* component into a sub-assembly while you are editing the top-level assembly, or while you are editing the sub-assembly in the context of a higher level assembly.

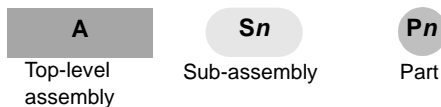
Right-click the sub-assembly icon in the FeatureManager design tree, and select **Open <sub-assembly.sldasm>** to open the sub-assembly in its own window. Then add a new component as described in **Creating a Part in an Assembly** on page 7-6.

## Forming a New Assembly from Existing Components

You can form a sub-assembly from components that are already in the assembly, thereby moving the components *down* one level in the assembly hierarchy.

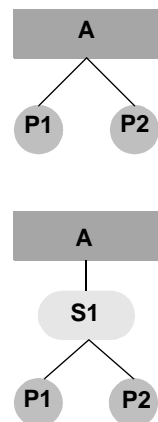
It is a good idea to position and mate at least one of the components before you begin, then select that component *first*. The origin of a newly formed sub-assembly is coincident with the origin of the *first* component that you select.

Throughout this section, the diagrams use these symbols:



### To form a new assembly from existing components:

- 1 In the FeatureManager design tree, select the components (individual parts or sub-assemblies) that you want to group into a sub-assembly. Hold the **Ctrl** key while you select.  
All the components must be at the same level within a single parent assembly. In this example, **P1** and **P2** are selected to form a new sub-assembly, **S1**.
- 2 Right-click one of the selected components, and select **Form New Assembly Here**.  
You can also select the components, then click **Insert, Component, Assembly from Selected Components**.  
The **Save As** dialog box appears.
- 3 Browse to a different folder if needed, enter a **File name**, and click **Save**. A new assembly document is saved in the folder you specify.



A new sub-assembly is inserted as the last component at the level where the selected components were located, and the components are moved into the new sub-assembly.

Instead of referencing the individual component documents **P1** and **P2**, assembly **A** now references the sub-assembly document **S1**.

Also, any mates that exist between components **P1** and **P2** move from the mategroup of assembly **A** to the mategroup of sub-assembly **S1**.

## Modifying a Sub-assembly

As you develop an assembly, you can modify its sub-assemblies in these ways:

- You can *dissolve* a sub-assembly into individual components, thereby moving the components *up* one level in the assembly hierarchy.
- You can *reorganize* an assembly by moving components up or down in the hierarchy, or to a different branch of the hierarchy.
- You can *change the order* of the components within a level of the hierarchy.

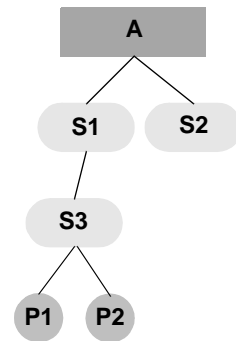
### IMPORTANT:

When you dissolve a sub-assembly, or reorganize the components at any level, the mates and any features that reference the selected components are affected. Therefore, decisions about hierarchical groupings should be made early in development of a complex assembly, in order to minimize the effects on these items.

- Mates move to the mategroup of the lowest common parent.*

For example, assume that there is a mate between **P1** and **P2**, in the mategroup of the nested sub-assembly **S3**.

- If you move **P1** up one level (into **S1**), the mate moves to the mategroup of **S1**, the lowest common parent on the same branch.
- If you move **P1** into **S2** (a different branch), the mate moves to the mategroup of the top-level assembly **A**, because **A** is the only common parent.
- If you dissolve **S3**, the mate moves to the mategroup of sub-assembly **S1**.



- Component patterns* have these specific limitations:
  - If you move the *seed* component of a pattern into a different assembly, the pattern feature and all the instances generated by the pattern are deleted.
  - If you dissolve a sub-assembly that has a component pattern, the pattern feature is removed, and the pattern instances become components of the assembly at the next higher level.
  - You cannot move individual instances generated by the pattern.

- ❑ *You cannot move a component with in-context features.* Any component that has external references to another component in the assembly cannot be moved into a different assembly.

If you move the *referenced* component, the references become out-of context. This applies to any part features created in the context of the assembly.

For example, assume that component **A** has a feature with an external reference to component **B**:

- You cannot move **A** into a different assembly.
  - If you move **B** into a different assembly, the external reference for the feature in **A** becomes out-of-context.
- ❑ *Parts created in context, including weld beads, cannot be moved into a different assembly.*
  - ❑ *Assembly feature cuts and holes are deleted.*
  - ❑ *Equations may not solve.* This is because, in many cases, the instance number suffix **<n>** of the components changes when you reorganize or dissolve an assembly. This is done automatically, to prevent conflicts with existing components in the destination assembly.
  - ❑ *Explode steps are deleted.*
  - ❑ *Routing sub-assemblies are subject to some special rules.* See **Reorganizing a Routing Sub-assembly** on page 13-19.

## Dissolving a Sub-assembly

You can dissolve a sub-assembly into individual components, thereby moving the components *up* one level in the assembly hierarchy.

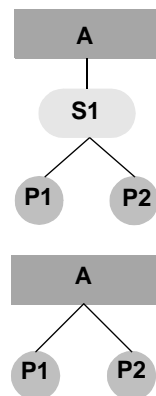
### **To dissolve a sub-assembly:**

In the FeatureManager design tree, right-click the sub-assembly you want to dissolve, and select **Dissolve Sub-assembly**. You can also select the sub-assembly icon and click **Edit, Dissolve Assembly**.

The components become components of the immediate parent assembly, and the sub-assembly is removed. The assembly document itself *is not deleted* from the folder where it is stored.

In this example, when you dissolve sub-assembly **S1**, components **P1** and **P2** become components of the top-level assembly **A**.

Instead of referencing the sub-assembly document **S1**, assembly **A** now references the individual component documents **P1** and **P2**.



---

**NOTE:** This is the reverse of the operation described in the section **Forming a New Assembly from Existing Components**.

---

## Reorganizing Assembly Components

You can move selected components up or down within their own branch of the assembly hierarchy, or move them to a sub-assembly in a different branch in the hierarchy.


There are two ways to reorganize components:

- Drag and drop the icons in the FeatureManager design tree.
- Use the **Reorganize Components** command on the **Tools** menu. This method can be easier to use when the FeatureManager design tree is very long, and therefore requires a lot of scrolling.

Whichever method you choose, all the components selected for a move operation must be from the same level of a single assembly.

### *To reorganize components in sub-assemblies (method 1):*

Drag the component icon in the FeatureManager design tree, and drop it on the icon for the destination assembly (either the top-level assembly, or a sub-assembly at any level of the hierarchy).

If the destination assembly is expanded, you can also drop the icon in the component section of the destination assembly. As you drag, watch for the  pointer.

To move multiple components at once, hold **Ctrl** as you select the icons.

If any components or features will be deleted as a result of the move, the **Reorganize Components** dialog box appears, with a list of the affected items.

The component icon appears at the end of the component section of the destination assembly.

If you are removing the last component of a sub-assembly, the empty sub-assembly is left in the FeatureManager design tree. If you want to move the last component up a level and remove the empty sub-assembly at the same time, dissolve the sub-assembly instead.

### *To reorganize components in sub-assemblies (method 2):*

- 1 Click **Tools, Reorganize Components**.
- 2 Select the components that you want to move, either in the FeatureManager design tree or in the graphics area. The selected components are listed in the **Components to move** box.
- 3 Click the **Destination assembly** box, then click the appropriate assembly icon in the FeatureManager design tree.

If any components or features will be deleted as a result of the move, the **Reorganize Components** dialog box expands to display a list of the affected items.


- 4 Click **Move**. The component icon appears at the end of the component section of the destination assembly.


## Changing the order of components

You can change the order of the components within a level of the assembly hierarchy.

By default, components appear in the FeatureManager design tree of an assembly in the order in which they are added. The same order is used in a bill of materials in a drawing of the assembly. By changing the order in the assembly, you can control the order in the bill of materials.

### **To change the order of components:**

Drag and drop the icons in the FeatureManager design tree to rearrange the order. The component is placed *below* the icon where you drop it (the highlighted icon). Watch for the  pointer.

To place a component directly below a sub-assembly, press **Alt** to change from a reorganize operation (with the  pointer) to a reorder operation.

## Editing a Sub-assembly in Context

You can edit a sub-assembly, at any level of the assembly hierarchy, in context. While you are editing a sub-assembly or one of its components, you can reference the geometry of surrounding components and features at any level of the hierarchy. You can also reference entities in a sub-assembly while you are editing the top-level assembly.

When you save the top-level assembly document, the edits that you have made to the sub-assembly in context are saved in the sub-assembly document.

### Limitations in Sub-assembly Editing

Most of the operations available for editing a sub-assembly on its own are available when editing a sub-assembly in context. However, there are a few operations that cannot be performed while editing a sub-assembly in context:

- creating a new component in context
- inserting or modifying a weld bead
- adding mates between components of the sub-assembly

For these operations, you must open and edit the sub-assembly in its own window.

### Automatic Change of Editing Focus

For some types of editing operations, the software recognizes that the changes you are making belong to the sub-assembly, and not to the top-level assembly. In those cases, the editing focus automatically changes from the top-level assembly to the sub-assembly.

If you edit any of the following items *that belongs to a sub-assembly*, the editing focus changes automatically: sketch, component pattern, mate, assembly feature (cut or hole), reference geometry.

Similarly, if you edit a feature (the definition or the sketch) of a sub-assembly component, the software automatically changes to **Editing Part** mode. References that you make while

editing a sub-assembly component are always made in the context of the top-level assembly, not the sub-assembly that contains it.

If you add a mate between a sub-assembly component and some geometry outside that sub-assembly (in the top-level assembly, or in another sub-assembly that is *not* nested under the active sub-assembly), the focus changes automatically, and the mate is added to the lowest level assembly that contains both components (the lowest common parent).


## Changing the Editing Focus Manually

For some types of editing operations, you need to specify that the changes you are making belong to the sub-assembly, and not to the top-level assembly. This includes any case where the edits could be successfully applied to either one. For example, you can add an assembly feature hole either to a sub-assembly, or to the top-level assembly, with different results.

You must activate the sub-assembly to perform these editing operations:

- Add a component from a file, or drag an existing part document from the Feature Palette, from an Explorer window, or from an open part document window into the sub-assembly. You cannot add a *new* component in the context of a sub-assembly.
- Delete a component from a sub-assembly.
- Add a component pattern, an assembly feature cut or hole, a sketch, or reference geometry.
- Add a mate between components of the active sub-assembly (both at the same level in the hierarchy, or one at a lower level, in a nested sub-assembly of the active sub-assembly).

---

**NOTE:** While a sub-assembly is active, the mates in the top-level assembly that position the sub-assembly with respect to the top-level are temporarily ignored. Top-level mates are solved again when you return to editing the top-level assembly and click **Rebuild** .

You cannot add a mate between a component *within* an active sub-assembly and a component *outside* the active sub-assembly.

---

- Use dynamic assembly motion on sub-assembly components. You can fix, float, and move sub-assembly components when the sub-assembly is active, to visualize the movement within the sub-assembly.

### ***To activate a sub-assembly for editing in context:***

Right-click the sub-assembly icon in the FeatureManager design tree, and select **Edit Subassembly**.

When you activate a sub-assembly, it turns pink, and the rest of the assembly turns gray, just as it does when you edit a part. When you edit a component of a sub-assembly, only the individual part being edited turns pink.

For more information about editing parts in the context of an assembly, see Chapter 7, “Working with Parts within an Assembly.”

## Exploding an Assembly View

---

For manufacturing purposes, it is often useful to separate the components of an assembly to visually analyze their relationships. Exploding the view of an assembly allows you to look at it with the components separated.

An *exploded view* consists of one or more *explode steps*.

Before you can explode an assembly, you must fully resolve any lightweight parts that it contains. See **Lightweight Parts** on page 6-40 and **Loading Additional Model Data** on page 6-41 for more information.

### Using AutoExplode

The AutoExplode function is useful when you need to create an exploded view of an assembly that has few components.

**To create an exploded view using AutoExplode:**

- 1 Click **Insert, Exploded View**.
- 2 In the **Assembly Explode** dialog box, click **AutoExplode**, and click **OK**.

The assembly explodes, and an **ExplView** feature is added to the active configuration.

---

**NOTE:** If the assembly does not explode as expected, try changing **Coincident** mates between faces to **Distance** mates with a value of zero. AutoExplode can always explode **Distance** mates; **Coincident** mates do not always explode automatically.







---

### Creating an Exploded View

**To explode an assembly:**

- 1 Click **Insert, Exploded View**.

The **Assembly Exploder** dialog box contains these **Step Editing Tools**:

- |   |  |
|---|--|
|  | <b>New</b> - create a new explode step                     |
|  | <b>Previous Step</b> - edit previous explode step          |
|  | <b>Next Step</b> - edit next explode step                  |
|  | <b>Undo</b> - undo all changes since the last <b>Apply</b> |
|  | <b>Delete</b> - delete current explode step                |
|  | <b>Apply</b> - apply/update current explode step           |



- 2 Click **New** . The **Assembly Exploder** dialog box expands.

Each area of the **Assembly Exploder** dialog box is automatically activated in the correct order needed to create an explode step. You do not need to click in any area of the dialog box to activate it unless you want to select and delete an incorrect entry.

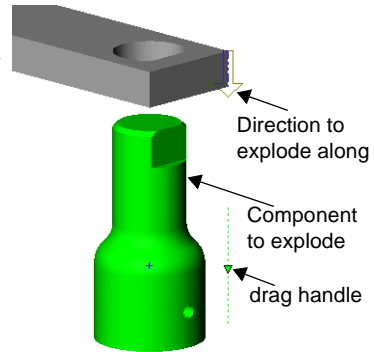
- 3 Click a component edge or face that is parallel to the direction you want to explode.


A preview arrow appears and a description of what you have selected appears in the **Direction to explode along** box.

If the preview arrow is pointing in the wrong direction, click the **Reverse direction** check box.

- 4 Click one or more components that you want to explode, either in the FeatureManager design tree or in the graphics area.


The names of the components are listed in the **Components to explode** box.




- 5 Click **Apply** . The component explodes from the assembly. Note the drag handle.

- 6 Drag the handle to position the component as desired.

- 7 Click **Apply**  to confirm this step.

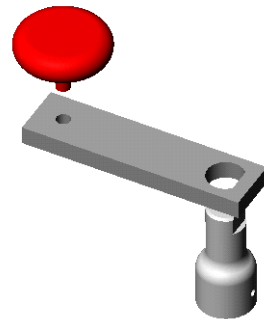
If you prefer to explode a component a precise distance, enter the value in the **Distance** box, then click **Apply** .

- 8 If you are satisfied with this exploded view, click **OK**.

If you want to add more explode steps to the exploded view, click **New** , repeat Steps 3 through 7, then click **OK** to finish the exploded view.

Remember to click **Apply**  after defining each step.



- 9 To collapse the assembly, right-click anywhere and select **Collapse**.

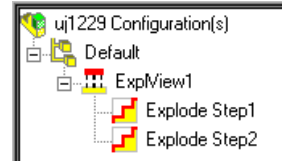


## Exploding and Collapsing an Exploded View

An exploded view is stored with the assembly configuration in which it is created.



### *To explode and collapse an exploded view:*

- 1 Click the Configuration tab  at the bottom of the window.
- 2 Click the  beside the desired configuration, and beside the **ExplView** feature to see the **Explode Steps**.
- 3 Double-click the **ExplView** feature. The assembly explodes.
- 4 To collapse the exploded view, right-click anywhere and select **Collapse**.




## Editing an Exploded View

### *To edit an exploded view (method 1):*

- 1 In the ConfigurationManager, expand the configuration so that you can select the **ExplView** feature.
- 2 Expand the **ExplView** feature to see the steps, then right-click the **ExplView** feature or the **Explode Step** you want to edit, and select **Edit Definition**.
- 3 In the **Assembly Exploder** dialog box, use the **Next Step**  and **Previous Step**  buttons to select each step in turn. You can also select a step from the **Explode steps** list.

As you select each step, the drag handle appears and the components involved are highlighted (**Shaded** view mode) or enclosed in bounding boxes (**Wireframe**, **Hidden In Gray**, or **Hidden Lines Removed** view mode).

- 4 Edit the explode step until the component is positioned as desired.
- 5 Click **Apply**  after editing each step.
- 6 Repeat for each explode step as needed, then click **OK** to finish the view.

### *To edit an exploded view (method 2):*

- 1 Double-click the **ExplView** feature to explode the view.
- 2 Right-click the component you want to reposition, and select **Show Explode Steps**. You can right-click the component in the graphics area, or you can switch to the FeatureManager, then right-click the component in the tree.

The drag handles for the explode steps on the selected component are displayed.

- 3 Drag the component by the green drag handle to the new position.
- 4 Repeat as needed for each component, then collapse the assembly.

## Detecting Interference Between Components

---

In a complex assembly, it may be difficult to visually determine whether components interfere with each other. You can determine the interference between components and examine the resulting interference volumes.

Before you can detect interference in an assembly, you must fully resolve any lightweight parts that it contains. See **Lightweight Parts** on page 6-40 and **Loading Additional Model Data** on page 6-41 for more information.

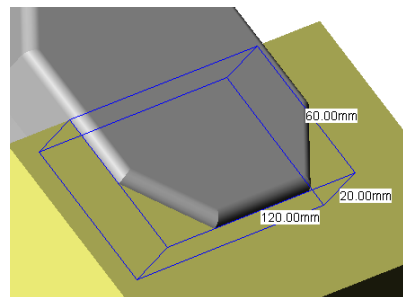
### *To check for interference between components in an assembly:*

- 1 Click **Tools, Interference Detection**.
- 2 Select two or more components in the assembly, or click an assembly icon (top-level or sub-assembly) in the FeatureManager design tree. If you click the top-level assembly, *all* the components in the assembly are checked for interference.
- 3 Click the **Treat coincidence as interference** check box if you want coincident entities (faces, edges, or vertices that touch or overlap) to be reported as interferences. Otherwise, touching or overlapping entities are ignored.
- 4 Click **Check**.

If there is interference, the **Interference results** box lists the interference occurrences (one occurrence is reported for each pair of interfering components). When you click an item in the list, the related interference volume is highlighted in the graphics area, and the names of the components involved are listed.

The volume of the interference is reported in the form of length, width, and height of the bounding box around the area of interference. These numbers are displayed in the graphics area.

- 5 With the dialog box still open, you can select other components to check for interference. Right-click in the graphics area and select **Clear Selections**, then select new components for checking, and click **Check**.
- 6 Click **Close** to dismiss the dialog box. When the dialog box is dismissed, the interference volumes are dismissed also.



---

**NOTE:** If detecting interference is important in your design work, check for interference each time you move or rotate a component.

---

## Simplifying Large Assemblies

---

Large assemblies can be complex, consisting of hundreds of components. You can simplify a complex assembly by toggling the *visibility* of the components, and by changing the *suppression state* of components.

There are several reasons to simplify an assembly:

- To improve system performance, and to reduce rebuild times, especially with very large assemblies.
- To create simplified views of the assembly that include certain components, and exclude others.
- To create design variations of the assembly, with different combinations of components, and different configurations of the components themselves.

### Toggling the Visibility

You can toggle the display of assembly components. Turning off the display of a component temporarily removes it from view, allowing you to work with underlying components more easily.


Hiding or showing a component affects *only the visibility* of the component in the active assembly configuration. Hidden components have the same accessibility and behaviors as shown components in the same suppression state (see the next section for information about suppression states).

Toggling the visibility does not affect the rebuild or evaluation speed. Display performance improves, however.

#### **To toggle the visibility of a component (part or sub-assembly):**

- 1 In the FeatureManager design tree or in the graphics area, right-click the component, and select **Component Properties**.  
To change several components at once, hold the **Ctrl** key as you select the components, then right-click and select **Component Properties**.
- 2 In the **Component Properties** dialog box, under **Visibility**, select the **Hide Model** check box, and click **OK**. The selected items are removed from the display.  
To show the components again, select the components in the FeatureManager design tree, then edit the **Component Properties**.

#### **Other ways to toggle the visibility:**

- Right-click the component, and select **Hide Component** or **Show Component**.
- Select the component, then click the **Hide/Show Component**  tool on the Assembly toolbar.
- Select the component, then click **Edit, Hide Component** (or **Show Component**).

## Suppression States

Depending on the scope of the work you plan to do at any given time, you can specify an appropriate suppression state for the components. This can reduce the amount of data that is loaded and evaluated as you work. The assembly displays and rebuilds faster, and you make more efficient use of your system resources.

There are two suppression states for assembly components: *resolved* and *suppressed*.

### Resolved

Resolved (or unsuppressed) is the normal state for assembly components. A fully resolved component is fully loaded in memory, fully functional, and fully accessible. All its model data is available, so its entities can be selected, referenced, edited, used in mates, and so on.

### Suppressed

You can use the suppressed state to remove a component temporarily from the assembly, without deleting it. It is not loaded in memory, and it is no longer a functional member of the assembly. You cannot see a suppressed component, or select any of its entities.

You can use this state to create multiple design variations (configurations) within a single assembly document. See **Creating a Configuration Manually** on page 8-2 for more information.

A suppressed component is removed from memory, so rebuild speed and display performance are improved. Because of the reduced complexity, evaluation of remaining components occurs faster.

Suppressing components also helps assemblies load quickly. However, mates that involve the suppressed components are suppressed also. As a result, the positions of components in the assembly can become under defined. In-context features that reference suppressed components may also be affected. When you restore a suppressed component to the fully resolved state, conflicts can occur. Therefore, use the suppressed state carefully when modeling.

## Lightweight Parts

You can load an assembly with its active parts *fully resolved* or *lightweight*. When a part is fully resolved, all its model data is loaded in memory. When a part is lightweight, only a subset of its model data is loaded in memory. The remaining model data is loaded on an as-needed basis.

## Advantages of Lightweight Parts

You can improve performance of large assemblies significantly by using lightweight parts. Loading an assembly with lightweight parts is faster than loading the same assembly with fully resolved parts.

Lightweight parts are efficient because the full model data for the parts is loaded only as it is needed. Only parts that you select, and parts that are affected by changes that you make in the current editing session, become fully resolved.

Assemblies with lightweight parts rebuild faster because less data is evaluated. Mates on a lightweight part are solved, and you can edit existing mates. In-context features that reference lightweight parts are solved.

### **To enable lightweight loading of parts:**


- 1 Click **Tools, Options, Performance**.
- 2 Under **Assemblies**, click the **Automatically load parts lightweight** check box.

---

**NOTE:** You can only change this option when no assemblies, or drawings of assemblies, are open.

---

When this option is selected, all parts are loaded lightweight when you open an assembly. The only exception is that any parts that are included in the feature scope of assembly feature cuts or holes are always loaded fully resolved (see **Working with Assembly Features** on page 7-16). Sub-assemblies themselves are not loaded lightweight, but the individual parts that they contain are lightweight.

When you point at a lightweight part in the graphics area, a feather pointer  is displayed, and the body is surrounded by a bounding box.

---

**NOTE:** Lightweight parts are only used when the model is in **Shaded** mode. If the model was last saved in **Hidden Lines Removed**, **Hidden In Gray**, or **Wireframe** mode, lightweight parts are not used.

---

## Loading Additional Model Data

When an operation requires model data that is not loaded for a lightweight part, the software fully resolves the part for you, to gain access to the additional data. This happens automatically when you do any of the following things:

- Click a part in the graphics area. When you click, the part becomes fully resolved, and the entity (face, edge, etc.) where you clicked is selected and highlighted.
- Right-click a part or feature, and select **Edit Part**, **Edit Sketch** or **Edit Definition**. The part becomes fully resolved, and you are in **Editing Part** mode.

- Rebuild an assembly that has in-context features, equations, or assembly features (cuts or holes) that affect a lightweight part.
- Create a drawing view of an assembly that contains lightweight parts. All lightweight parts become fully resolved in the assembly when the view is added to the drawing sheet.

Certain global operations require that many parts be fully resolved at once:

- **Mass Properties** (see page 5-25)
- **Interference Detection** (see page 6-38)
- **Exploded View** (see page 6-35)
- **Advanced Show/Hide, Advanced Selection** (see page 6-47)
- **Section view** of the assembly model (see page 5-23)
- editing equations that involve lightweight parts
- exporting to other file types (IGES, SAT, and so on. See Chapter 14, “Importing and Exporting Files.”)
- displaying non-shaded view modes

To specify what you want to happen when you request one of these operations, click **Tools, Options, Performance**, then choose one of these options under **Resolve lightweight parts**:

- **Always.** All lightweight parts are fully resolved for you, and they are included in the operation.
- **Prompt** (default). A dialog box appears, asking if you want the software to resolve the lightweight parts for you. Click **Yes** to fully resolve the parts and continue, or click **Cancel**. If you also click **Always resolve** (before you click **Yes** or **Cancel**), the option is set to **Always**.

---

**NOTE:** When a lightweight part becomes fully resolved, all instances of the part, in any configuration, in any open assembly documents, become fully resolved.

Any part that is resolved during the session stays resolved until you close the assembly. All the parts are loaded lightweight again the next time you open the assembly, as long as the option is still selected.

---


## Changing the Suppression State

You can change the suppression state of any component (part or sub-assembly) to the **Resolved** or **Suppressed** state at any time.

### *To change the suppression state of components:*

- 1 In the FeatureManager design tree or the graphics area, right-click the desired component, and select **Component Properties**.  
To change several components at once, hold the **Ctrl** key as you select the components, then right-click and select **Component Properties**.
- 2 In the **Component Properties** dialog box, under **Suppression State**, select the desired state, and click **OK**.

### *Other ways to change the suppression state:*

- Click a component, then either click **Change Suppression State**  on the Assembly toolbar, or click **Edit, Suppress** (or **Unsuppress**, to resolve the component).
- To fully resolve (unsuppress) a suppressed component, right-click the suppressed component, and select **Set to Resolved**.
- To fully resolve all lightweight parts, right-click the assembly icon at the top of the FeatureManager design tree, and select **Set Lightweight to Resolved**.
- Suppress or resolve all the components at once by opening the assembly in a new configuration. See **Special Assembly Configurations** on page 8-6.



**Table 6-4. Comparison of component suppression states**






	<b>Resolved</b>	<b>Lightweight</b>	<b>Suppressed</b>	<b>Hidden</b>
loaded in memory	yes	partially	no	yes
visible	yes	yes <sup>2</sup>	no	no
features available in FeatureManager design tree	yes	yes <sup>3</sup>	no	no
faces and edges accessible for adding mates	yes	yes <sup>3</sup>	no	no
mates solved	yes	yes	no	yes
in-context features solved	yes <sup>1</sup>	yes <sup>4</sup>	no	yes
assembly features solved	yes	yes <sup>4</sup>	no	yes
considered in global operations	yes	on demand <sup>2</sup>	no	yes
may be edited in context	yes	yes <sup>3</sup>	no	no
load and rebuild speed	normal	faster	faster	normal
display speed	normal	normal	faster	faster

<sup>1</sup> In-context features on *resolved* parts that *reference* lightweight parts update automatically.

<sup>2</sup> Global operations include Mass Properties, Interference Detection, Exploded View, Advanced Selection and Advanced Show/Hide, solving equations, displaying non-shaded views and section views, and exporting to other file formats. When lightweight parts are present, the software prompts you to either resolve them, or cancel the operation.

<sup>3</sup> Lightweight parts are resolved automatically when selected for this operation.

<sup>4</sup> When in-context features, assembly features, or equations affect lightweight parts, the lightweight parts are resolved when the assembly is rebuilt.

 Part1<1>	<b>Resolved.</b> This is the only suppression state where the component can be expanded (double-click the name, or click the  ).
 Part1<1>	<b>Lightweight.</b> A feather overlay appears on the icon of a lightweight part.
 Part1<1>	<b>Suppressed.</b> A gray icon indicates that the component is not in use in the active configuration.
 Part1<1>	<b>Hidden.</b> A white icon indicates that the component is active, but invisible.

## Assembly Envelopes

---

An *assembly envelope* is a special type of assembly component. An envelope is not a *functional* component of the assembly. It is a *reference* component, and it is ignored in global assembly operations (Bill of Materials, Mass Properties, and so on). This functionality allows you to select components based on their positions with respect to the envelope volume (inside, outside, or crossing).

You can use an envelope by itself to quickly change the visibility of assembly components. You can also use an envelope (either alone or in combination with file properties), to select components for other editing operations, such as suppress, copy, or delete.

There are two ways to insert an envelope component:

- Insert an existing part file to use as the envelope.
- Create a new envelope component in the context of the assembly.

In either case, the envelope component must be a *solid* volume (not shelled). This is because selection with envelopes is based on *interference* between the volumes of the functional assembly components and the envelope component. It is *not* based on the *containment* of components within the bounding box of the envelope component.

Envelope components are displayed in a light blue transparent color when you use **Shaded** view mode. An **Envelope** feature is added to the ConfigurationManager for each envelope component that you create. Also, envelope components are identified in the FeatureManager design tree.

### ***To create an assembly envelope from a file:***

- 1 Click **Insert, Envelope, From File**.
- 2 In the **Open Reference** dialog box, select the part file to use as an envelope.
- 3 Click in the assembly window where you want to place the envelope component. You can mate the envelope component to the other components in the assembly to position it precisely.

### ***To create an assembly envelope in the assembly context:***

- 1 Click **Insert, Envelope, New**.
- 2 In the **Save As** dialog box, enter a name for the new envelope component. A part document with this name is created.
- 3 Click a face or plane in the assembly window where you want to begin sketching the envelope component. **Plane1** of the new envelope component is mated to the selected face or plane with an **Inplace** mating relation.
- 4 Create the base feature and any additional features of the envelope component.

You can reference the geometry of the assembly components as you create the envelope component. If any of the referenced entities changes, the envelope updates accordingly.

- 5 When you are done defining the new envelope component, right-click anywhere in the graphics area, and select **Edit Assembly:assembly\_name**.

### *To remove an assembly envelope:*

- 1 Select the envelope component in either the FeatureManager design tree or the graphics area. You can also select the **Envelope** feature in the ConfigurationManager.
- 2 Press the **Delete** key. Both the envelope component and the **Envelope** feature are deleted.

### *To use the envelope for selection:*

- 1 In the ConfigurationManager, right-click the **Envelope** feature, and choose **Select using envelope**.
- 2 Choose one or more criteria for selecting the components:
  - **Inside envelope.** Components that are completely inside the envelope are selected.
  - **Outside envelope.** Components that are completely outside the envelope are selected.
  - **Crossing envelope.** Components that cross the boundary of the envelope are selected.
- 3 Click **OK**. Components that meet the criteria are selected, and you can perform the desired operation (**Suppress**, **Hide**, **Delete**, and so on).

### *To use the envelope for component visibility:*

- 1 In the ConfigurationManager, right-click the **Envelope** feature, and choose **Show/Hide using envelope**. The **Apply Envelope** dialog box appears.
- 2 Specify the envelope criteria for selecting the components (see the previous procedure).
- 3 Click **Show part components** or **Hide part components** to specify the visibility of components that meet the criteria.
- 4 Specify what to do with any component that does not meet the criteria:
  - If you clicked **Show part components** in Step 3, you can either **Hide it** (regardless of current display state), or **Leave its show/hide state as is**.
  - If you clicked **Hide part components** in Step 3, you can either **Show it** (regardless of current display state), or **Leave its show/hide state as is**.
- 5 Click **OK**. The display is updated to hide or show components according to your selections.

## Component Selection by Properties

---

You can select components for editing operations based on properties. Properties include those specified under **File, Properties, Custom** and **Configuration specific**, as well as some properties that are specific to SolidWorks documents, including **Part volume, Configuration name, Document name, and Part mass**.

You can specify a set of properties and values to be used as selection criteria. You can combine properties using the logical operators **And** and **Or**.

For example, you assign the custom property *System* to each component in an assembly. You specify the name of the system the component belongs to as the value. Then you select all components that have the custom property *System* whose value is *Cooling Or Heating*. This selects all the components that belong to either the cooling system or the heating system of the assembly.

You can save any set of selection criteria as a *query*. Then you can reload it whenever you want to apply the same set of criteria again. The query (**.sqy**) is an ASCII text file that you can read and edit using a text editor.


Before you can use these component selection methods in an assembly, you must fully resolve any lightweight parts that it contains. See **Lightweight Parts** on page 6-40 and **Loading Additional Model Data** on page 6-41 for more information.

### Advanced Component Selection

You define the selection criteria by specifying the properties, conditions, and property values of components you want to select.

#### *To define advanced selection criteria:*

- 1 Click **Tools, Advanced Select**.
- 2 Under **Define additional criteria**, select a **Property** from the list. Properties include those listed under **File, Properties**, on the **Custom** or **Configuration specific** tabs, plus several special SolidWorks properties.
- 3 Select a **Condition**, and enter a **Value**.
  - For most properties, conditions include standard mathematical operators.
  - For some properties, conditions include **is (exactly)**, **is not**, and **contains**. You use **Contains** to specify a text string which is part of the text of the value; for example, the value “simplified” *contains* the string “sim”.
  - For the SolidWorks special property **Part volume**, conditions include **is inside**, **is crossing**, and **is outside**. The values that are available are the names of any envelopes that you have defined. See **Assembly Envelopes** on page 6-45.
- 4 Click the **Add** button to include the selection criteria in the **Criteria** list.

- 5 To add another criteria to the list, click **And** or **Or** as a logical operator for combining the criteria, then specify another **Property**, **Condition**, and **Value**. Click **Add** after specifying each new criteria.
- 6 To delete a criteria from the list, click in the *first column* of the **Criteria** list box, *beside* the criteria to delete. A blue box appears beside the selection. Click **Delete**.
- 7 When you have specified all the necessary criteria, click **Apply**. The selected components are highlighted. If needed, you can modify the selection, and click **Apply** again.
- 8 If you want, you can **Save Criteria** for future use, or **Load Criteria** that you saved earlier. See **Saving and Reloading Selection Criteria** on page 6-49.
- 9 Click the pushpin  if you want to keep the dialog box open for further selections and operations.
- 10 Perform the desired editing operation, such as **Suppress**, **Hide**, or **Copy**.

### Advanced Show/Hide Components

For show and hide operations, you can also control the visibility of components that *do not meet* the specified criteria. You use the same methods to select the components as described in the previous section, **Component Selection by Properties** on page 6-47.

#### *To define advanced visibility criteria:*

- 1 In the ConfigurationManager, right-click the active configuration, and select **Advanced Show/Hide**.
- 2 Define the selection criteria by specifying the properties, conditions, and values of components you want to select.
- 3 Under **Operations**, click **Show part components** or **Hide part components** to specify the visibility of the components that meet the criteria.
- 4 Specify what to do with any component that does not meet the criteria:
  - If you clicked **Show part components** in Step 3, you can either **Hide it** (regardless of current display state), or **Leave its show/hide state as is**.
  - If you clicked **Hide part components** in Step 3, you can either **Show it** (regardless of current display state), or **Leave its show/hide state as is**.
- 5 Click **Apply**. If needed, you can modify the selection, and click **Apply** again.
- 6 If you want, you can **Save Criteria** for future use, or **Load Criteria** that you saved earlier.
- 7 Click **Close** when you are satisfied with the selections.

## Saving and Reloading Selection Criteria

When you use **Advanced Selection** or **Advanced Show/Hide**, you can save the selection criteria to a file. Then you can load the criteria later, to quickly apply the same set of selection criteria whenever you want.

### **To save selection criteria:**

- 1 In the **Advanced Selection** or **Advanced Show/Hide** dialog box, specify a set of selection criteria.
- 2 Click **Save Criteria**.
- 3 In the **Save Criteria** dialog box, browse to the folder where you want the file to be stored, enter a file name, and click **Save**.

The query is saved with the extension **.sqy**

### **To reload selection criteria:**


- 1 In the **Advanced Selection** or **Advanced Show/Hide** dialog box, click **Load Criteria**.
- 2 In the **Load Criteria** dialog box, browse to and select the query file (**.sqy**), and click **Open**. The query is loaded, and the criteria are listed in the dialog box.

## Customizing the Appearance of an Assembly


---

By default, components that you add to an assembly are displayed using the color specified in the original part document. You can override the part color for selected instances, or use the default color for the assembly. You can also modify material properties such as **Transparency** and **Shininess**. These changes do not affect the original part document.

### **To change the color of selected component instances:**

- 1 Click a component in the FeatureManager design tree or in the graphics area. To select multiple components, hold **Ctrl** as you select.
- 2 Click **Edit Color**  on the Standard toolbar.
  - To select a new color, choose one from the palette, or define a custom color.
  - To use the default color for the *assembly* document, click **Use assembly** (see the next procedure).
  - To restore the component to the original color specified in the *part* document, click **Remove**.
- 3 Click **OK**.

**To specify a default color for the assembly:**

- 1 Click the assembly icon at the top of the FeatureManager design tree, then click **Edit Color** .
- 2 Choose a color from the palette, or define a custom color, and click **OK**. This is the color that is used when you click **Use assembly** for a selected component.

**To edit the color and advanced properties of selected component instances:**


- 1 Right-click the component instance in the graphics area or in the FeatureManager design tree, and select **Component Properties**.  
To change the color of several components at once, hold **Ctrl** while you select the components, then right-click, and select **Component Properties**.
- 2 Click the **Color** button.
  - To apply a new color, click **Change Color**, choose a color from the palette, and click **OK**.
  - To use the default color for the *assembly* document, click **Use Assembly**.
  - To restore the component to the original color specified in the *part* document, click **Remove Color**.
- 3 Click **Advanced** to set other properties for the selected component, such as **Transparency** and **Shininess**.
- 4 Click **OK**. The color of the selected instances changes, but not the referenced part document.

**To change the color of all instances of a component, including the original part document:**

- 1 Right-click the component in the graphics area or in the FeatureManager design tree, and select **Edit Part**.
- 2 Click **Tools, Options, Color**.
- 3 Select **Shading**, and click **Edit**.
- 4 Select a new color, click **OK** to close the **Color** dialog box, and **OK** again to close the **Options** dialog box.
- 5 To return to editing the assembly, right-click the assembly icon at the top of the FeatureManager design tree, and select **Edit Assembly**.

## Displaying Annotations in an Assembly

---

To specify the display of annotations in an assembly, right-click the **Annotations** folder  in the FeatureManager design tree, and select **Details**. In the **Annotation Properties** dialog box, you have these options:

- **Display filter.** Select which annotation types, if any, to display by default. To select specific types, clear the **Display all types** check box, and select the types you want.
- **Text scale.** Adjust the size of the text as needed.
- **Always display text at the same size.** When checked, all annotations and dimensions are displayed at the same size, regardless of zoom.
- **Display items only in the view in which they are created.** When checked, any annotation is displayed only when the model is viewed in the same orientation as when the annotation was added.
- **Display assembly annotations.** Toggles the display of annotations, including annotations that belong to the assembly, and annotations that are displayed in the individual part documents.
- **Use assembly's setting for all components.** Sets the display of *all* annotations to match the setting for the assembly document, regardless of the setting in the individual part documents. Use this option along with **Display assembly annotations** to display different combinations of annotations.

For complete information about creating and using annotations, see Chapter 10, “Detailing.”

## Assembly Statistics

---

You can generate a set of statistics for an assembly document. The report includes such items as the total number of components, the number of unique parts and sub-assemblies, the number of components in each suppression state, the number of top-level mates, and the maximum depth of the assembly hierarchy (for nested sub-assemblies).

To generate a list of assembly statistics, click **Tools, Assembly Statistics**. When you are finished reading the report, click **OK** to close the dialog box.



## Sending Assembly Documents

---

You can send assembly documents by electronic mail. You can choose to send only the assembly itself, or you can send the assembly and all referenced documents.

This function uses the electronic mail application installed on your system.

### *To mail an assembly file:*

- 1 With an assembly document active, click **File, Send To**.
- 2 In the **Send Mail** dialog box, select one of the following:
  - **Assembly\_name.sldasm only**
  - **Assembly\_name.sldasm and other documents it references.**

A list of the referenced documents, the number of documents, and the combined size of the documents is displayed.

---

**NOTE:** The size of the combined documents may be important to you because mailing large size documents may cause transmission problems.

---

If you choose to send the assembly and referenced documents, you do not have to send all the documents. You can:

- Click a filename you do not want to send, and click **Remove**.
  - Click a removed filename and click **Attach** to replace the file.
  - Click **Attach All** to return all the removed files.
- 3 Click **OK** to send the documents.
  - 4 Enter your mail password, the mailing address, and any other information requested by your mail application.

## Working with Parts within an Assembly

---

In this chapter, you learn about *top-down* assembly design, including several techniques for building and modifying components within the context of an assembly.

These techniques are based on *external references* and *time-dependent features*.


- ❑ *Create and edit components within the context of the assembly.* That way, you can use the geometry of one component to help define another. When you create references between components, the models are fully associative. Changes you make to a referenced component cause the others to update accordingly.
- ❑ Start with a *layout sketch* that defines fixed component locations, planes, and so on, then design the parts referencing these definitions.
- ❑ Create cut or hole features that are added only after the components are assembled. These *assembly features* exist only in the context of the assembly.
- ❑ Place *patterns of components*, either by referencing a pattern feature on one of the components, or by creating a pattern in the assembly.
- ❑ Create *molds* of various types.
- ❑ Merge the bodies of assembly components by using a *join* operation.
- ❑ Add *weld beads* between assembly components.

## External References

---

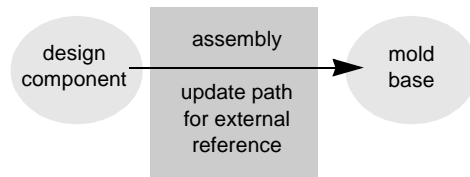
An *external reference* is created when one document is dependent on another document for its solution. If the referenced document changes, the dependent document changes also.

In an assembly, you can create an in-context feature on one component that references the geometry of another component. This in-context feature has an external reference to the other component. If you change the geometry on the referenced component, the associated in-context feature changes accordingly.

Updating of in-context features occurs automatically if the update path is available. The update path is contained in the assembly where you create the reference. Update paths for in-context features are displayed at the bottom of the FeatureManager design tree with this icon . If the update path is not available (for example, if the assembly is closed), updating occurs the next time you open the assembly that contains the update path.

Consider the example of an assembly that contains two components: a mold base and a design component.

In the context of the assembly, you create a cavity feature in the mold base, with an external reference to the design component. The update path for the cavity feature is therefore contained in the assembly.



Now suppose that you need to change the design component, and update the mold base accordingly.

- If the assembly is *open*, the cavity feature in the mold base updates automatically when you make the changes to the design component.
- If the assembly is *closed*, the cavity feature cannot be updated to reflect the changes until you open the assembly, thereby making the update path available.

In the FeatureManager design tree, any item with an external reference has a suffix that indicates the status of the reference:

- The suffix **->** means that the reference is *in-context*. It is solved and up-to-date.
- The suffix **->?** means that the reference is *out-of-context*. The feature is not solved or not up-to-date. To solve and update the feature, open the assembly that contains the update path.

### **To list the external references on a part or feature:**

Right-click the part or the feature with the **->** or **->?**, and select **List External References**. The referenced components, features, and entities are listed, as well as the status of each reference.

---

**NOTE:** A component cannot have an external reference to itself (a circular reference).

You cannot reference one instance of a part to create a feature on another instance of the same part, even if the instances are in different configurations.

You cannot replace a part with another part created by deriving the second part from the first. See **Derive Component Part** on page 5-17 for more information.

---

## Setting External Reference Options

*To set external reference behavior:*

- 1 Click **Tools, Options, External References**.
- 2 Choose from these options:
  - **Open referenced documents with read-only access.** Specifies that all referenced documents will be opened for read-only access by default. If you have read-only access to a document, another user can open, edit, and save that document. Then you can reload the referenced document when you want to update your document.
  - **Don't prompt to save read-only referenced documents (discard changes).** Specifies that when this document is saved or closed, no attempt will be made to save its referenced documents that are opened for read-only access.
  - **Allow multiple contexts for parts when editing in assembly.** Allows you to create external references for a single part in more than one assembly context. However, any individual *feature* or *sketch* can only have external references *in one assembly*.
  - **Load referenced documents.** Lets you specify whether to load referenced documents when you open a part that has external references (such as a derived part, or a part that has been edited in the context of an assembly).

If you click **Never**, referenced documents are never opened. This can cause the references to be out-of-date until you open the referenced documents.

If you click **Always**, referenced documents are always opened, in a minimized window.

If you click **Prompt**, you are asked about loading externally referenced documents each time you open a part with external references. Click **Yes** or **No** in the dialog box that appears when you open the part. If you also click **Don't ask me again**, the option is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

If you do not open the referenced documents, the external references are considered out-of-context, and are marked with **->?**. To resolve the out-of-context reference, open the referenced document, so that changes can be propagated as needed.

- 3 Choose from these options under **Assemblies**:
  - **Automatically generate names for referenced geometry.** When this check box is *cleared* (the default), you can mate to parts for which you have read-only access. Unless you plan to replace components, leave this check box *cleared*, especially in a multi-user environment. See **Preserving Mates with Component Replacement** on page 6-16 for more information.
  - **Update component names when documents are replaced.** Clear this check box if you use component names in the FeatureManager design tree that are different from the filenames of the components. See page 6-13 for information about component names.
- 4 Click **OK**.

### Searching for Referenced Documents

When you open any parent document, the other documents that are referenced in the parent document are loaded in memory also. In the case of assemblies, components are loaded in memory according to the suppression state they were in when the assembly was saved. By default, the software searches for referenced documents *in the following order*:

- 1) The paths specified in the **Folders** list, if the **Search document folder list for external references** check box is selected. (See the procedure that follows.)
- 2) The last path you specified to open a document.
- 3) The last path the system used to open a document (in the case of the system opening a referenced document last).
- 4) The path where the referenced document was located when the parent document was last saved.
- 5) The path where the referenced document was located when the parent document was last saved *with the original disk drive designation*.

For example, assume that you had stored documents on the **C:** drive, then moved them to the **E:** drive. Step 4 searches the proper directories on the **E:** drive, and Step 5 searches the proper directories on the **C:** drive.

- 6) If a referenced file still is not found, you are given the option to browse for it.

---

**NOTE:** All updated reference paths in the parent document are saved when you save the parent document.

---

#### **To use a document folder list to search for external references:**

- 1 Click **Tools, Options, External References**.
- 2 Select the **Search document folder list for external references** check box to turn on the search behavior. (This lets you define the folder list, then turn the search behavior on or off without affecting the contents of the list.)

- 3 Under **Show folders for**, select **Documents**, then choose from these options:
  - To *add* a path to the search order, click **Add**. The **Choose Directory** dialog box appears. Browse to the desired directory, and click **OK** to add the path to the search list for **Documents**.
  - To *delete* a path from the search list, select the path in the list, and click **Delete**.
  - To *change* the search order, select a path in the list, and click **Move Up** or **Move Down** to change the path's position in the list.
- 4 Click **OK**.

---

**NOTE:** You can also specify the paths for **Palette features**, **Palette parts**, **Palette forming tools**, and **Custom symbols** in this section.

Search paths for these items are *always* used, whether the **Search document folder list for external references** check box is selected or not. See **Organizing Feature Palette Items** on page 11-8 for more information.

---

## Saving Assemblies with In-context Features

There are some special considerations when you save an assembly with in-context features. Parts that have features defined in the context of an assembly must maintain the association with the parent assembly in order to update correctly, both inside and outside of the assembly context.

If you need two versions of the same part (one that includes the in-context features, and one that does not include them), you must edit the referenced file locations when you save the assembly. This prevents the in-context features from affecting the original part document.

Consider this example. In a mold assembly, you edit the mold base in context to create a cavity. If you want to keep the original, unedited mold base part for use in other mold assemblies, you need to do one of the following:

- Save the assembly and the parts to a different folder, and edit the references to reflect the new path.
  - or –
- Save the edited mold base part with a new name, and replace the part in the assembly.

When you save the assembly, you save its updated reference to the edited mold base (*with* the cavity). This mold base updates correctly, if the update path is available when you make changes (see the example on page 7-2). The original mold base is left unchanged.

See also **Replacing a Component in an Assembly** on page 6-12 for more information.

For general information about saving assembly documents, see **Saving an Assembly and Its Parts** on page 6-10.

## Editing Parts in the Assembly Context

---

While editing an assembly, you can:

- Create a new part in the assembly window
- Edit an existing component in the assembly window
- Open an existing component document in its own window

Whenever you are creating or editing a component (part or sub-assembly) in the context of an assembly, the component and its name in the FeatureManager design tree are displayed in pink. This is to make it obvious that you are editing the *component*, not the *assembly*.

To change the color, click **Tools, Options, Color**. Select the **View system defaults** check box, then select a new color for **Edit Part in Assembly** (in the **System** list).


## Creating a Part in an Assembly

You can create a new part in the context of an assembly. That way you can use the geometry of other assembly components while designing the part. The new part has its own part file so you can modify it independently from the assembly.

You can also create a new sub-assembly in the context of the top-level assembly. See **Creating a Sub-assembly** on page 6-28 for more information.

### ***To create a part within an assembly:***

- 1 In an assembly document, click **Insert, Component, New Part**.
- 2 In the **Save As** dialog box, enter a name for the new part and click **Save**. The new part is saved in its own document so you can edit it separately.

The name of the new part appears in the FeatureManager design tree, and the component pointer  appears.

- 3 If the assembly is empty, select a plane from the FeatureManager design tree. Otherwise, select a plane or planar face on which to position the new part.

A sketch is automatically opened in the new part. An **InPlace** (coincident) mate is added between **Plane1** of the new part and the selected plane or face.


The new part is fully positioned by the **Inplace** mate. No additional mates are required to position it. If you wish to reposition the component, you need to delete the **Inplace** mate first. See **To create smart mates while moving a part:** on page 6-26.

- 4 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.

---

**NOTE:** If you extrude a feature using the **Up To Next** option, the *next* geometry must be on the same part. You cannot use the **Up To Next** option to extrude to a surface on another component in the assembly.

---


- 5 Click **File, Save**, then select the part name in the **Resolve Ambiguity** dialog box, or select the assembly name to save the entire assembly and its components.
- 6 To return to editing the assembly, right-click the assembly name in the FeatureManager design tree, or right-click anywhere in the graphics area, and select **Edit Assembly:assembly\_name**, or click **Edit Part** .

## Editing a Part in an Assembly

Editing a part while in an assembly allows you to modify a component without leaving the assembly. You can also reference surrounding geometry while creating new features. The sketches that you create while editing a part in the assembly can use any edge or can be dimensioned to any edge of any part. Features may use other end conditions, such as **Up to Surface**, to another component. See **Specifying End Conditions** on page 4-4.

You can also edit a sub-assembly in the context of the top-level assembly. See **Editing a Sub-assembly in Context** on page 6-33 for more information.

### *To edit a part while in an assembly:*

- 1 Right-click the part in the FeatureManager design tree or the graphics area, and select **Edit Part**, or click **Edit Part**  on the Assembly toolbar.

The title bar shows the name of the part in the assembly that is open for editing as **part\_name in assembly\_name.sldasm**. Note that the message in the status bar indicates that you are now editing the *part* document even though the entire *assembly* is visible.


---

**NOTE:** When you edit a part in the context of an assembly, the referenced configuration in the assembly becomes the active configuration in all open windows (for example, if the part is open in its own window).

If the part is used in another open assembly document, with the **Use component's in-use or last saved configuration** option selected, those instances are also affected.

For more information, see **Component Configurations in an Assembly** on page 8-7.

---

- 2 Make the necessary changes to the part. You can make most of the same types of changes in the context of an assembly as you can make in the part's own document.
- 3 To return to editing the assembly, right-click the assembly name in the FeatureManager design tree, or right-click anywhere in the graphics area, and select **Edit Assembly:assembly\_name**, or click **Edit Part**  again.



## Opening Documents from Part or Assembly Windows

While you are editing an assembly, you can open its associated component documents (parts or sub-assemblies) in their own windows. Any changes that you make to the components automatically update the assembly.

While you are editing a part with in-context features, you can open the assembly document where the features were created.

### ***To open a component from an assembly document:***

- 1 In the *assembly* window, right-click a component in the FeatureManager design tree or graphics area.
- 2 Select **Open *part\_name.sldprt*** (or ***sub-assembly\_name.sldasm***).  
The component document opens in a separate window.

### ***To open an assembly from a part document:***

- 1 In the *part* window, right-click a feature that has an external reference (with the suffix -> or ->?).
- 2 Select **Edit In Context**.  
The assembly that contains the update path for the selected feature opens in a separate window. See **External References** on page 7-2 for more information.

## Time-Dependent Features

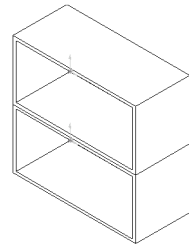
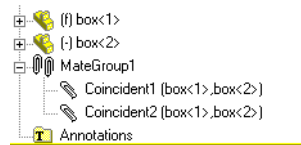
Time-dependent features are the basis of top-down design. They include:

- ❑ Assembly features (assembly feature cuts or holes, and weld beads)
- ❑ Components that are created in the context of the assembly
- ❑ In-context features, reference geometry, and sketches that reference other components
- ❑ Component patterns

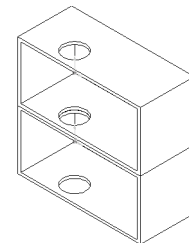
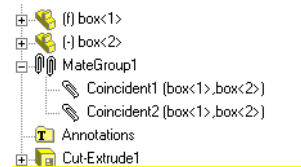
When a time-dependent feature references components that have already been positioned by mates, the features become dependent on both the components and the mates that position the components. To view these dependencies, right-click the feature and select **Parent/Child**.

Mates are solved together as a system. This system is called a *mategroup*. Each assembly has a default mategroup, **MateGroup1**. Within a mategroup, the order in which mates are added does not matter; all the mates are solved at the same time. In a bottom-up design, only one mategroup is required

In this example, coincident mates are used to align the faces of the components.

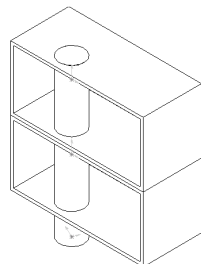
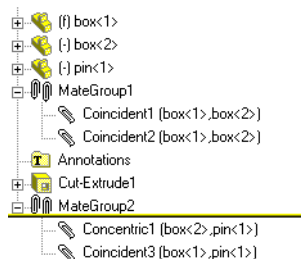


Then a cut is extruded through both components. This is an *assembly feature*, not a cut feature in the part document.



Now add a component (a pin, for example) to the assembly.

If you mate the pin to the assembly feature cut (which is time-dependent), an additional mategroup is required. This is because the new component can only be positioned after the time-dependent feature has been updated.



If a component is referenced in more than one mategroup, it is treated as a fixed component in all but the first mategroup.

Additional mategroups are added automatically when required and removed automatically when the last mate they contain is deleted. You cannot add or delete mategroups manually. When a new mate is added, it is put in the earliest mategroup possible, based on parent/child relationships.

---

**NOTE:** It is good design practice to only reference time-dependent features for mating when that is the only way you can achieve the design intent of the assembly. You can edit the positions of the components with more flexibility when time-dependent features are not involved, because the order of evaluation does not matter.

---

It is possible to have a situation where two components have been partially positioned in **MateGroup1**, and in order to complete the positioning, you need to add a mate which references a time-dependent feature that is dependent on **MateGroup1**.

In the example earlier in this section, you could do this by mating the flat face of the pin coincident to the top face of the upper part first. Then add the assembly hole, and add the concentric mate between the pin and the assembly hole. The coincident mate is moved to the second mategroup for a successful solution. This is because it must be solved simultaneously with the concentric mate to position the pin, and the concentric mate can only be solved after the assembly hole is updated.

In this example, the software detects that the coincident mate needs to move forward, and moves the mate when the concentric mate is performed. However, in more complicated cases you may need to determine which mates to move forward. To do this, delete the mates that position the component you wish to move, then add the mate to the time-dependent feature (to create the additional mategroup), then add mates as needed to fully position the component.

When positioning a component to geometry that depends on a time-dependent feature, it is good design practice to mate components to the time-dependent feature *first*, then add additional mates as needed.

## Adding a Component Pattern

---

You can define a pattern for placing components in an assembly in much the same way as you define a feature pattern in a part.

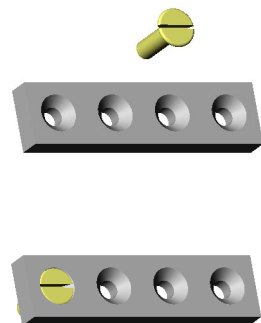
You can also place a pattern of components in an assembly based on a feature pattern of an existing component. For example, you can insert a set of bolts in a pattern of bolt holes on a part.

### ***To define a new pattern for placing components in an assembly:***

- 1 Insert and mate components in an assembly.
- 2 Click **Insert, Component Pattern**.
- 3 In the **Pattern Type** dialog box, click **Define your own pattern (Local)**, choose **Linear** or **Circular**, and click **Next**.
- 4 Select the components to copy either in the FeatureManager design tree or in the graphics area. The components are listed in the **Seed component(s)** box.
- 5 Click the **Along Edge/Dim** box, select an entity to indicate the pattern direction, and click **Reverse Direction** if necessary.
  - For a linear pattern, select a linear edge or a linear dimension.
  - For a circular pattern, select an axis, a linear edge, or an angular dimension.
- 6 Specify the **Spacing** and **Instances** (the total number of instances, including the seed components).
- 7 If you want to create a pattern in two directions (linear patterns only), change **First Direction** to **Second Direction**, and repeat Steps 5 and 6.
- 8 Click **Finish**.

### ***To use an existing feature pattern to place a pattern of components in an assembly:***

- 1 Insert into the assembly a component with a pattern feature, and one or more components to mate with the pattern.
- 2 Mate the components to be copied to the *original* feature used to create the pattern on the first component (not one of the copied instances of the feature).
- 3 Click **Insert, Component Pattern**.
- 4 In the **Pattern Type** dialog box, click **Use an existing feature pattern (Derived)** and click **Next**.
- 5 Select the components to copy either in the FeatureManager design tree or in the graphics area. The components are listed in the **Seed component(s)** box.



- 6 Click the **Pattern Feature** box, then click either the pattern feature in the FeatureManager design tree, or one of the pattern instances in the graphics area.
- 7 Click **Finish**.



***To remove an instance in a component pattern:***

- 1 Expand the pattern feature in the FeatureManager design tree, and select the instance you want to delete,  
– or –  
Click a face on the instance you want to delete in graphics area.
- 2 Press the **Delete** key.

***To restore the instance:***

- 1 Right-click the pattern feature in the FeatureManager design tree, and select **Edit Definition**.
- 2 In the **Positions to Skip** box, select the instance and press the **Delete** key (to remove it from the list).

By default, all the component instances generated by a pattern use the same configuration as the original component. To use a different configuration for a selected instance, edit the **Component Properties** of the instance. See **Component Configurations in an Assembly** on page 8-7.

See **Linear Pattern** on page 4-41 and **Circular Pattern** on page 4-42 for more information about patterns.

## Reorder and Roll Back

---

### Reorder

Changing the order of assembly items in the FeatureManager design tree is dictated by parent/child relationships, just as it is with features in parts.

- You can reorder sketches and reference geometry that belong to the assembly.
- You can reorder components within their own level of the assembly hierarchy. See **Changing the order of components** on page 6-33.
- You can reorder mates within their own mategroups.
- You cannot reorder mategroups, component patterns, in-context features, or any default assembly items (default planes, **Annotations** or **Lighting** folders, etc.).

See also **Reorganizing Assembly Components** on page 6-32.

### Roll Back

You can roll back reference geometry, sketches, component patterns, assembly feature cuts and holes, mategroups and in-context features. In general, rolling back assembly items behaves the same as rolling back features in parts. The following behaviors are specific to assemblies:

- If you roll back before a mategroup, all components positioned by that mategroup are also rolled back. They are removed from the display until you roll forward again.
- If you roll back an assembly pattern, the components generated by that pattern are rolled back.
- You cannot roll back into the section of the FeatureManager design tree that lists the components (that is, after some of the components are added, or before others). This is because components are not time-dependent. Hide or suppress the components instead.

## Using an Assembly Layout Sketch

You can design an assembly from the top-down using layout sketches. You can construct one or more sketches showing where each assembly component belongs. Then, you can create and modify the design before you create any parts. In addition, you can use the layout sketch to make changes in the assembly at any time.

The major advantage of designing an assembly using a layout sketch is that if you change the layout sketch, the assembly and its parts are automatically updated. You can make changes quickly, and in just one place.

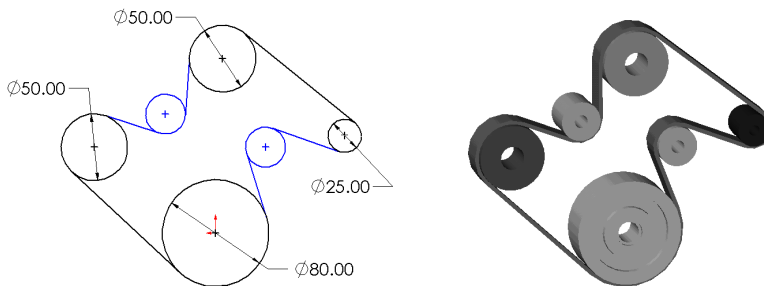
To use an assembly layout sketch, do the following:

- ❑ Create a layout sketch in which various sketch entities represent parts in the assembly. Indicate a tentative location for each component, capturing the overall design intent.
- ❑ Reference the geometry in the layout sketch when you create each component. Use the layout sketch to define the component size, shape, and location within the assembly; make sure that each part references the layout sketch.

### ***To design an assembly using the layout sketch technique:***

The following example is typical of an automobile engine accessory drive belt system. The system has several pulleys and idlers, connected by a serpentine belt.

- 1 Create an assembly sketch. Use circles to represent the outer rims of the pulleys. Use tangent lines between the circles to represent the serpentine belt. Dimension the size and location of each pulley.
- 2 For *each* pulley, insert a new part, sketch a circle, add a **Coradial** relation with the corresponding circle in the layout sketch, then extrude the sketch. Add other features to the pulley parts as needed.
- 3 Using the layout sketch as a guide, create the belt around the pulleys as another new part.



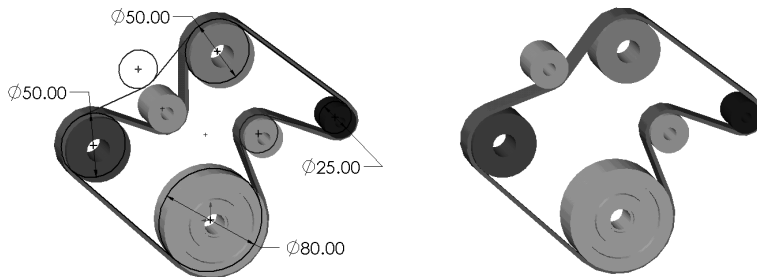
You can change the location or the size of the pulleys by editing the layout sketch.

The assembly now consists of a belt and a set of pulleys whose locations and sizes are driven by the layout sketch. The advantage of this type of design is clear when you have to make changes to the design.

For example, working *in the layout sketch*, you can:

- Rearrange the pulleys so that a slipping pulley has more of its surface covered by the belt.
- Make a driven pulley larger so less torque is required to turn it.
- Drag the pulleys and belt to dynamically experiment with different locations.
- Set an angle dimension to specify how much of each pulley must be covered by the belt.

When you exit the layout sketch, the assembly and the parts are updated.



If you designed this assembly without a layout sketch, you would need to change many dimensions, or move components in the assembly, then rebuild each time.

The layout sketch does not have to be the master plan for a design. If you had a model of an engine with some fixed pulley locations, you could make the circles in the layout sketch coincident with the known locations of the pulleys in the model. You would still have the freedom to drag the other pulleys to different locations, and the assembly would update automatically when you made changes.



## Working with Assembly Features

---

While in an assembly, you can create cut or hole features that exist *in the assembly only*. You determine which parts you want the feature to affect by setting the scope.

This is useful for creating cuts or holes that are added *after* the components are actually assembled, and that affect more than one component. When you want to add a cut or hole to a single component in an assembly, it is better to edit the part in context than to use an assembly feature.

While it is not a requirement, it is good practice to fully define the positions of the components of the assembly, or fix their locations before you add assembly features. This helps prevent unexpected results if the components are moved later.

### Specifying the Scope

The *scope* of an assembly feature is the list of parts that are affected by the feature. The default scope includes *all* the parts in the assembly. However, you can specify a scope that includes only certain components that you select.

#### **To set the scope:**

- 1 In an assembly, click **Edit, Feature Scope**.
- 2 Select the components to include. You can click the components either in the graphics area or in the FeatureManager design tree. The list of selected components is displayed in the **Edit Assembly Feature Scope** dialog box.

To remove a component from the list, click the component again. To remove all the components from the list, right-click in the graphics area, and select **Clear Selections**.

- 3 Click **OK**.

### Creating an Assembly Feature

Assembly feature cuts and holes affect the assembly only; the individual part files are not affected. You can set the scope before you create an assembly feature, or you can edit the scope afterwards.

#### **To create an extruded cut assembly feature:**

- 1 Open a sketch on a face or plane, and sketch a profile of the cut.
- 2 Click **Insert, Assembly Feature, Cut, Extrude**.
- 3 Set the **Depth**, **Type**, and other options as needed in the **Extrude Cut Feature** dialog box. See **Specifying End Conditions** on page 4-4.
- 4 Click **OK**.

***To create an assembly feature revolved cut:***

- 1 Open a sketch on a face or plane, and sketch a profile of the cut and a centerline around which the profile revolves.
- 2 Click **Insert, Assembly Feature, Cut, Revolve**.
- 3 Specify the **Type** and **Angle** in the **Revolve Feature** dialog box.
- 4 Click **OK**.

***To create an assembly feature hole:***

- 1 Click the planar face approximately where you want to create the hole.
- 2 Click **Insert, Assembly Feature, Hole, Simple**.
- 3 Specify the **Diameter** and **Type**. If you choose the **Blind** type, specify the **Depth**. You can also specify **Draft** if needed.
- 4 Click **OK**.
- 5 To locate the hole precisely, right-click the **Hole** feature, and select **Edit Sketch**. Dimension the sketch as needed.

***To edit an assembly feature:***

Right-click the assembly feature in the FeatureManager design tree, and select either **Edit Sketch**, **Edit Definition**, or **Feature Scope**.

## Creating Molds

A mold consists of a base containing a cavity with the shape of the desired part. When you create a mold you must account for the *scaling factor*, that is the amount the material in the mold shrinks or expands as it solidifies. The scaling factor depends on the type of material used and the shape of the mold, and is expressed as a percentage (+/- 20%) of the linear size (not volume) of the design part.

The software scales the cavity by the specified scaling factor, using this formula:

$$\text{cavity size} = \text{part size} * \left( 1 + \frac{\text{scaling factor}}{100} \right)$$

For example:

Desired adjustment	Scaling factor in % (enter this number)	Resulting cavity size
2% material shrinkage (cavity larger than part)	2.0	part size * 1.02
2% material expansion (cavity smaller than part)	-2.0	part size * 0.98

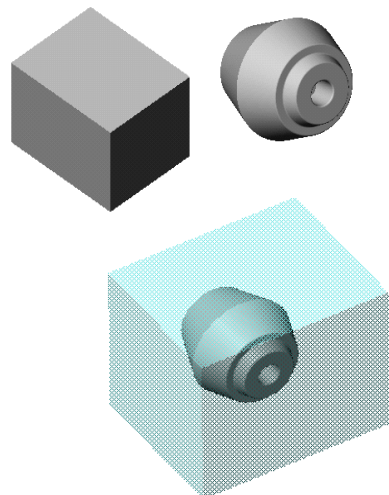
### Creating Drafts for Molds

You may need to add draft angles to molded design parts, to make it possible to remove the finished part from the mold. Refer to **Split Line** on page 3-12 and **Draft** on page 4-29 for information about splitting faces and adding draft angles.

## Simple Mold

### To create a mold cavity:

- 1 Create the design part you want to mold.
- 2 Create the mold base. Make sure that the mold base is large enough to contain the part you want to mold.
- 3 Create an assembly containing the mold base and one or more design parts. You can insert multiple design parts in a single mold base to create multiple cavities.
- 4 Position the design part *inside* the mold base, using mating relationships as needed.



- 5 In the assembly window, right-click the mold base in the graphics area or in the FeatureManager design tree and select **Edit Part**,

– or –

Click the mold base, then either click **Edit Part**  on the Assembly toolbar or click **Edit, Part**.

---

**NOTE:** It is important to be aware that you are editing the *part*, not the *assembly*, because the changes you are about to make will be reflected in the original part file of the mold base.

If you use standard mold base sizes, and do not want the originals to be affected by this operation, use the **Save As** command in the mold base part document to save it with a different name for use in each new mold assembly. Otherwise, the original mold base will include the cavity you are about to insert.

Also see **Editing Referenced File Locations** on page 6-14 for information about ways to replace the references to the mold base component.

---

- 6 Click **Insert, Features, Cavity**. The **Cavity** dialog box appears.
- 7 In the FeatureManager design tree, select the design parts. The components are listed in the **Design Component** box.
- 8 Enter the **Scaling Factor in %** (to a maximum of 20%).  
A positive value expands the cavity, a negative value shrinks the cavity. See the examples at the beginning of this section.
- 9 Select the point about which scaling occurs in the **Scaling Type** list.
  - **About Component Centroids**. Scales the cavity for each part about its own centroid.
  - **About Component Origins**. Scales the cavity for each part about its own origin.
  - **About Mold Base Origin**. Scales the cavity for each part about the origin of the mold base part.
- 10 Click **OK**.

---

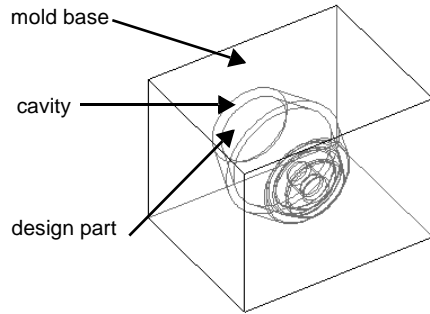
**NOTE:** If there are multiple solutions for creating the cavity (if creating the cavity results in more than one separate solid body), you must choose which solution to keep.

In the **Resolve Result Ambiguity** dialog box, click the **Next** and **Previous** buttons to see the possible solutions. The highlighted geometry will be *kept* (not *removed*) when the cavity is created. Click **OK** when the solution that you want is highlighted.

---

A cavity in the shape of the design part is created. The cavity size reflects the scaling factor you specified.

Any changes you make to the design part automatically update the cavity in the mold base, as long as the update path is available (see **External References** on page 7-2).





### **To cut the mold base to make two mold halves:**

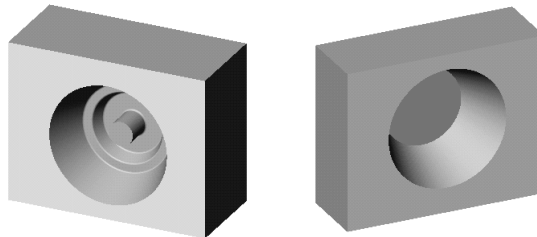
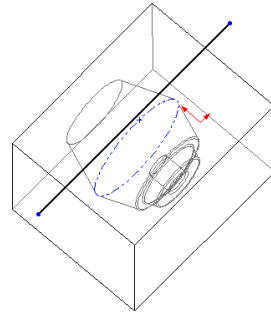
- 1 In the FeatureManager design tree or the graphics area, click the mold base and select **File, Derive Component Part**.

A part file is created for the derived part (a part that has another part as its first feature).

- 2 Open a sketch on a planar face of the mold base.
- 3 Sketch a line that, when projected, divides the mold base as desired (the parting line).

You can also select a model edge, and click **Convert Entities**  on the Sketch Tools toolbar to project the edge as a parting line.

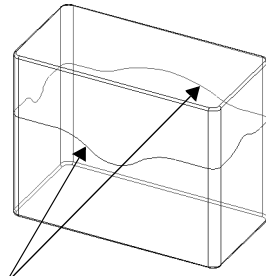
- 4 Click **Extruded Cut**  on the Features toolbar, or click **Insert, Cut, Extrude**.
- 5 In the **Extrude Cut Feature** dialog box, specify a **Type** of **Through All**, and leave the **Flip Side to Cut** check box cleared.
- 6 Click **OK**. The mold base is cut, exposing the cavity within the mold base. This is one half of the mold.
- 7 Save this part.
- 8 To create the other half of the mold, save the part again with another name, then edit the definition of the cut. To reverse the direction, select the **Flip Side to Cut** check box.



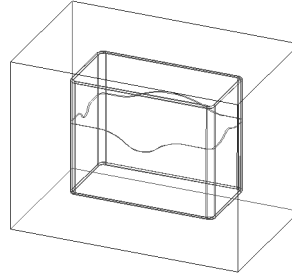
## Mold with Complex Parting Lines

Some types of design parts require complex parting lines. When an extruded cut cannot produce the desired parting line, you can use a radiated surface to cut the mold.

- 1 Create a design part with split lines (or model edges) on the faces where you want to split the mold. See **Split Line** on page 3-12.
- 2 Following the procedure for a **Simple Mold** on page 7-18, create an interim assembly, insert a cavity in the mold base, and create a derived part for each half of the mold.

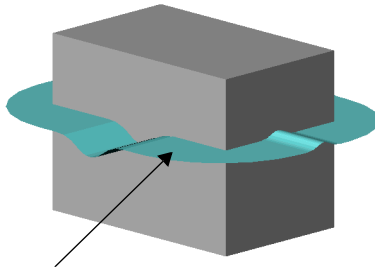


Use a **Split Line** to create edges for cutting the mold.

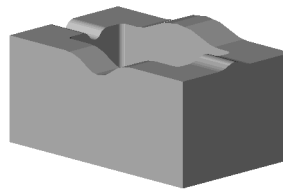


Interim assembly, with a cavity

- 3 In each derived part, insert a **Radiate Surface**. Select the edges created by the split line (or the model edges), and specify a **Radiate distance** that is large enough to extend the surface beyond the outside faces of the mold base. See **Radiate Surface** on page 4-53.
- 4 Select the surface, and click **Insert, Cut, With Surface**. In one of the derived mold halves, click **Flip the side to cut away** when you insert the cut. See **To cut a part with a surface**: on page 4-57.



Create a **Radiate Surface** that extends beyond the faces of the mold base.



Finished mold half, with surface hidden

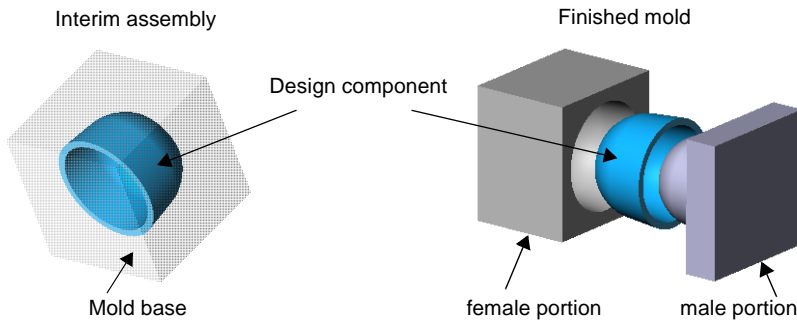
## Core and Cavity Mold

In the **Simple Mold** on page 7-18, the base of the design part was extruded from a midplane, with an inward draft angle. By parting the mold at the midplane, the design part can easily be removed from the mold.



In this example, the design part is a bowl with drafted sides. To remove the design part from the mold, the parting line needs to be coincident with the top edge of the bowl. A portion of one mold piece fits inside the other piece. You need to cut the mold differently to create the necessary pieces.

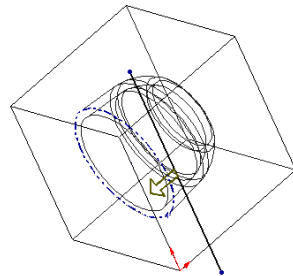
To prepare the parts for cutting, follow the instructions beginning on page 7-18:

- 1 Create an interim assembly containing the mold base and the design part. Mate the components as needed.
- 2 Create the cavity in the mold base part.
- 3 Derive two component parts: one for the female portion of the mold and one for the male portion.



### ***To create the female portion of the mold:***

- 1 In one of the derived parts, open a sketch on a side of the mold base that is perpendicular to the top face of the bowl.
- 2 Select the edge of the cavity that corresponds to the top edge of the bowl, and click **Convert Entities** . Drag the ends of the converted line beyond the edges of the mold base.
- 3 Click **Extruded Cut**  on the Features toolbar, or click **Insert, Cut, Extrude**. Specify a **Type** of **Through All**. If the preview arrow is not pointing away from the bowl, click **Flip side to cut**, and click **OK**.



The **Resolve Result Ambiguity** dialog box appears. This lets you choose which piece of the model to keep when there are multiple solutions.






To see each of the solutions, click the **Next** and **Previous** buttons. The highlighted geometry will be *kept* (not *removed*) when the feature is created.

When the cut removes the material above the cavity, there are two possible pieces left:

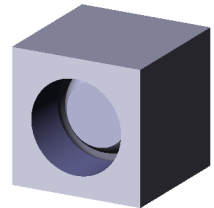
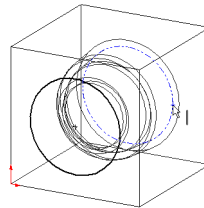
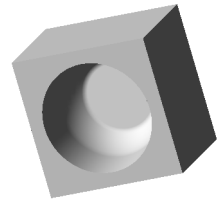
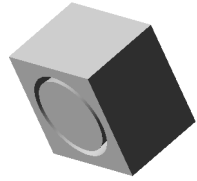
- A dome-shaped piece (the material inside of the cavity). Reject this solution.
- A box with a cup-shaped indentation (the material outside of the cavity). Select this solution.

- 4 Click **OK** when the solution that you want is highlighted.

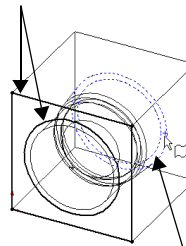
**To create the male portion of the mold:**

- 1 In the other derived part, open a sketch on the face of the mold base that is closest to the *bottom* of the bowl. Select the *inside* edge at the *top* of the cavity, and click **Convert Entities** .
- 2 Click **Extruded Cut** . Specify a **Type** of **Up to Next**, and click **OK**. The **Next** faces are the faces of the cavity; this cut exposes those faces. Now you can see the shape on the inside of the mold base.
- 3 Open another sketch on the same face.
  - Select the model face you are sketching on again, and click **Convert Entities**  to project the outside edges of the mold base into the sketch.
  - Select the *outside* edge at the *top* of the cavity, and click **Convert Entities** .
- 4 Click **Extruded Cut**  again. Specify a **Type** of **Up to Surface**, then click the face at the *top* of the cavity. Click **OK**.

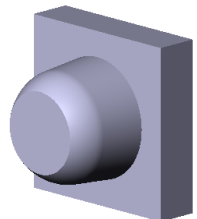
The cut removes all the material *between* the sketch loops, up to the selected face.



Convert entities to create two sketch loops



Extrude up to this face





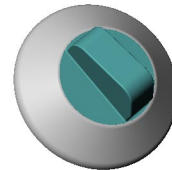
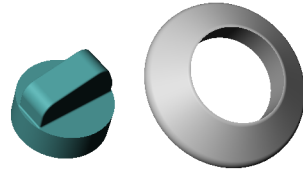
## Joining Parts

You can join two or more parts to create a new part. The join operation removes surfaces that intrude into each other's space, and merges the part bodies into a single solid volume.

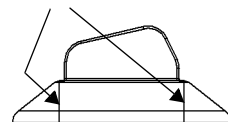
### To join parts:

- 1 Create the parts that you want to join, then create an assembly containing the parts.
- 2 Position the parts as desired in the assembly. The parts may either touch each other or intrude into one another.
- 3 **Save** the assembly but do not close the window.
- 4 Insert a new part into the assembly:
  - a) Click **Insert, Component, New Part**.
  - b) Enter a name for the new part and click **Save**.
  - c) Click a plane or planar face on a component on which to position the new part.
- 5 Click **Insert, Features, Join**. The **Join** dialog box appears.
- 6 In either the FeatureManager design tree or the graphics area, click the components you want to join. The names are listed in the **Design component** box.
- 7 Select the **Force surface to surface contact** check box if you want to join any coincident faces (as well as intruding volumes).
 

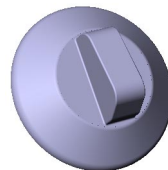
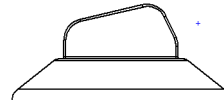
The software finds the affected faces, then extends the face with the smallest area into the other component, and fills in any resultant gaps.
- 8 Select the **Hide input components** check box to hide the original components after the join is complete. See **Toggling the Visibility** on page 6-39 for information about hidden components.
- 9 Click **OK** to create the joined part.
- 10 To exit from **Editing Part** mode, right-click in the graphics area, and select **Edit Assembly:assembly\_name**.



Before Join - coincident faces



After Join - faces removed, bodies merged



### To identify which parts are used to create a joined part, either:

- Select the joined component, and click **File, Find References**.
- Expand the joined component, then expand the **Join** feature to see the absorbed references.

**To edit the definition of the joined part to add or remove components:**

- 1 In the FeatureManager design tree, right-click the **Join** feature of the joined part and select **Edit Definition**.  
– or –  
In the graphics area, right-click the joined component and select **Edit Definition**.
- 2 Click the parts that you want to remove from the joined component, or click additional parts to be joined to the existing component.  
To remove all the parts, right-click in the graphics area, and select **Clear Selections**.  
The components that make up the joined component are listed in the **Design Component** box.
- 3 Click **OK**. The joined component updates to reflect the changes.

---

**NOTE:** The joined part is fully associative to the original parts and to the assembly. Any changes made to the original parts are reflected in the joined part. If the parts move relative to each other in the assembly, the joined part changes also.

---

## Welding

You can add a variety of weld types to an assembly. Table 7-1 lists the weld types that are supported for the ISO and ANSI dimensioning standards.

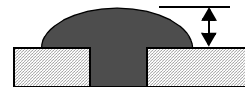
**Table 7-1. Weld types**

ISO	ANSI
Butt	Square
Square Butt	Scarf
Single V Butt	V Groove
Single Bevel Butt	Bevel
Single V Butt with Root	U Groove
Single Bevel Butt with Root	J Groove
Single U Butt	Flare-V
Single J Butt	Flare-Bevel
Backing Run	Fillet
Fillet	Seam
Seam	Back or Backing
	Flange-Edge
	Flange-Corner

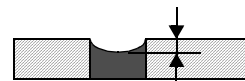
### Surface Delta and Radius

When you add a weld bead with a **Surface Shape** of **Convex** or **Concave**, you must specify the **Top Surface Delta**. For a backing run weld, you also specify the **Bottom Surface Delta**. If the **Surface Shape** is **Flat**, there is no surface delta.

For a convex weld, the top surface delta is the distance between the highest point on the bead and the contact faces.



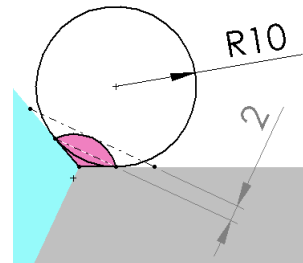
For a concave weld, the distance is measured *below* the top surface rather than *above*.



For a fillet weld, you also specify the **Radius**.

The radius is measured as the radius of an imaginary ball rolling along the seam. The edges of the weld bead are at the points of tangency, where the ball meets the contact faces.

In this fillet weld, the specified **Radius** is 10mm, and the **Top Surface Delta** is 2mm.



## General Procedure for Welding Components

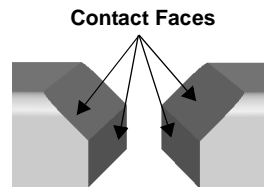
### To add a weld bead between components:

- 1 Insert the components in an assembly, and add mates to position them as needed for the specific weld type you plan to use.
- 2 Click **Insert, Assembly Feature, Weld Bead**.
- 3 In the **Weld Bead Type** dialog box, select a weld **Type** from the list.
- 4 Click **Next**.
- 5 In the **Weld Bead Surface** dialog box:
  - Select a **Surface Shape** from the list (**Flat**, **Convex**, or **Concave**).
  - Specify the **Top Surface Delta**, the **Bottom Surface Delta** and **Radius** as required for the **Type** and **Surface Shape** selected.
- 6 Click **Next**.

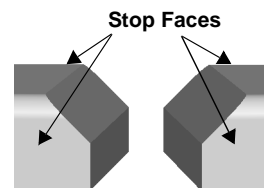
The **Weld Bead Mate Surfaces** dialog box appears. Be sure to read the information in the dialog box about selecting faces for the weld type you have chosen.

- 7 Select the necessary faces. For all welds, you must select **Contact Faces**. Depending on the type of weld you are adding, you may also need to select **Stop Faces** and **Top Faces**. Click each box, then select the faces in the graphics area.

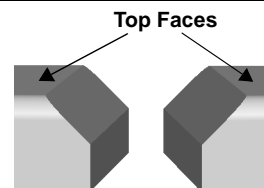
**Contact Faces** are the faces that are joined by the weld bead. Select a face (or a set of adjacent faces) on each component.



**Stop Faces** are the faces that define the beginning and the end of the weld bead. On each component, select two faces (or two sets of adjacent faces) that share an edge with the contact faces.



**Top Faces** are the faces from which to measure the **Top Surface Delta**. On each component, select one face that shares an edge with a contact face.



- 8 Click **Next**.
- 9 In the **Weld Bead Part** dialog box, either accept the default name for the **Weld Bead Part**, or enter a new part name, and click **Finish**.




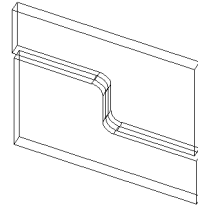
**To edit a weld:**


- 1 In the FeatureManager design tree, expand the **Bead** component.
- 2 Right-click the **Weld Bead** feature and select **Edit Definition**.
- 3 In each of the weld bead dialog boxes, change the parameters as needed. Click **Next** to continue, then click **Finish** when you reach the last dialog box.

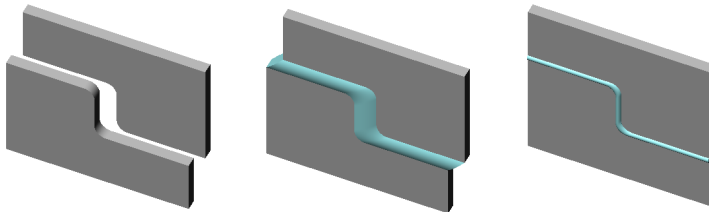
## Face Selection Tips

When components have complex shapes, or when the mates place the components very close to each other, selecting the faces for a weld can be difficult. There are several ways you can make it easier:

- Change the view to **Wireframe** or **Hidden In Gray**.
- Rotate the model. You can change the orientation while you select the faces.
- Set the **Selection Filter** to **Filter Faces** .
- Temporarily add some space between the components:
  - Edit the values of **Angle** and **Distance** mates.
  - Suppress one or more mates, then move the components apart slightly.



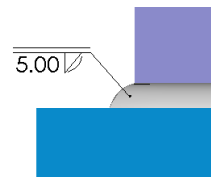
After you complete the weld bead, edit or unsuppress the mates as needed to restore the components to the correct positions. Then click **Rebuild**  to update the weld bead.



## Weld Symbols

When you create a weld, a weld symbol is automatically attached to the weld bead component in the assembly.

For information about importing a weld symbol to a drawing, see **Importing Model Annotations** on page 10-18.



**To add more detailed specifications to the default weld symbol:**

- 1 In the assembly document or in a drawing document, right-click the symbol, and select **Properties**, or double-click the symbol.
- 2 Specify the dimensions, symbols, and other options that you want to include, and click **OK**. See **Weld Symbol** on page 10-39 for more information.

## Working with Configurations

---

Configurations allow you to create multiple variations of a model within a single document.

- ❑ In part documents, configurations allow you to create families of parts with different dimensions, features, and properties.
- ❑ In assembly documents, configurations allow you to create simplified versions of the design by suppressing or hiding components. You can also create families of assemblies using different configurations of the components, different parameters for assembly features, or different dimensions.

You can create configurations manually, or you can use a design table to create multiple configurations simultaneously.


Design tables provide a convenient way to create and manage configurations in an easy-to-use worksheet. You can use design tables in both part and assembly documents, and you can display design tables in drawings.

## Creating a Configuration Manually

---

To create a configuration manually, you specify its properties, then modify the model to create the necessary variations in the new, active configuration.

### To create a configuration manually:

- 1 In either a part or assembly document, click the Configuration tab  at the bottom of the FeatureManager design tree to change to the ConfigurationManager.
- 2 In the ConfigurationManager, right-click the part or assembly name and select **Add Configuration**. The **Add Configuration** dialog box appears.
- 3 Enter a **Configuration Name**. The name must not include the forward slash (/) or at (@) characters.
- 4 Add **Comments** if desired.
- 5 Select or clear the **Properties for newly inserted items** options as needed.

These options control what happens to *this* configuration when you add new items to *another* configuration. The options that are available depend on the type of document.

	Option cleared	Option selected
<u>Part documents</u>		
<b>Suppress features</b>	New features are included (not suppressed) in this configuration.	New features are suppressed in this configuration.
<u>Assembly documents</u>		
<b>Suppress features and mates</b>	New features* and mates are included (not suppressed) in this configuration.	New features* and mates are suppressed in this configuration.
<b>Suppress component models</b>	New components are resolved (not suppressed) in this configuration.	New components are suppressed in this configuration.
<b>Hide component models</b>	New components are shown in this configuration.	New components are hidden in this configuration.

\* This includes assembly feature cuts and holes, component patterns, reference geometry, and sketches *that belong to the assembly* (not to one of the assembly components).

- 6 In an assembly document, select the way properties are applied to sub-assemblies:
  - Click **Apply properties to sub-assembly root only** to apply the properties selected in Step 5 to only the top-level sub-assembly and part components. All sub-assembly child components inherit the properties specified in the configuration of the sub-assembly.
  - Click **Apply properties to ALL sub-assembly components** to apply the properties selected in Step 5 to all added components. For example, if you want all

components of newly added sub-assemblies to be suppressed, regardless of the properties specified in the sub-assembly configuration, select this option.

- 7 Click **Advanced** to specify how a configuration is referenced when it is used in a bill of materials.

**Part No. displayed when used in a Bill of Materials.** Select one of the following:

- **Use document name.** Select this option (the default) if you want the document name to be used in the part number column in a bill of materials.


If multiple configurations of the part are used in an assembly, all instances are grouped as a single item number.

- **Use name below.** Select this option to specify a different value for the part number column in a bill of materials. The configuration name is filled in by default. Alternatively, you may enter a different name (for example, a stock number).

If multiple configurations of the part are used in an assembly, each *unique* name is listed as a separate item number. (Multiple configurations can use the same name.)

**Don't show child components in BOM when used as sub-assembly** (assembly documents only). Select this option if you *never* want this configuration to be expanded in the bill of materials, when it is used as a sub-assembly. Otherwise, the child components may be listed in the bill of materials when certain options are selected. See **Bill of Materials (BOM)** on page 10-31.

Click **OK** to return to the **Add Configuration** dialog box.

- 8 Click **OK** to create the new configuration.
- 9 Click the FeatureManager tab  to return to the FeatureManager design tree.
- 10 Modify the model as needed to create the design variation. See the next section for a list of items that are supported in configurations.
- 11 Save the model.

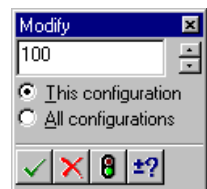
## Configurable Items

Part configurations support the following design variations:

- Modify feature dimensions.

When a part has multiple configurations, the **Modify** box lets you specify where to apply the changed dimension: to **This configuration** only, or to **All configurations**.

- Suppress features in the active configuration. See **Suppressing Features** on page 5-13.
- Define configuration-specific properties. See **Custom Properties** on page 5-21.





Assembly configurations support the following design variations:

- Change the suppression state (**Suppressed, Resolved**) or visibility (**Hide, Show**) of components.
- Change the referenced configuration of components.
- Change the dimensions of distance or angle mates, or suppress unwanted mates.
- Modify the dimensions or other parameters of features that belong to the assembly. This includes assembly feature cuts and holes, component patterns, reference geometry, and sketches *that belong to the assembly* (not to one of the assembly components).
- Suppress features that belong to the assembly.
- Define configuration-specific properties. See **Custom Properties** on page 5-21.

---

**NOTE:** If you want to modify a dimension value of a feature for a selected configuration, you must use one of these methods:

- Use a design table to control the value.
- Double-click the feature to display the dimension. Then double-click the dimension, change the value in the **Modify** box, and select **This configuration**.


If you modify the dimension value by editing the *definition* of the feature, the value is modified for ALL configurations.

This applies in *part and assembly* documents, to all types of feature dimensions, including the dimensions of solid features, the values used in mates (distance or angle), the number of instances in patterns, and so on.

---

## Activating a Configuration

**To activate a different configuration:**

- 1 Click the Configuration tab  to change to the ConfigurationManager.
- 2 Right-click the name of the configuration you want to view and select **Show Configuration**.

– or –

Double-click the configuration name.

The named configuration becomes the active configuration, and the view of the model updates to reflect the newly selected configuration.

## Editing a Configuration

---

### **To edit a configuration:**

- 1 View the desired configuration.
- 2 Change to the FeatureManager view, then:
  - In a part document, change the suppression state of features or the dimensions as needed.
  - In an assembly document, change the suppression state or visibility of components, component configurations, and so on, as needed.

### **To edit the configuration properties:**

- 1 Right-click the configuration name, and select **Properties**.
- 2 Edit the name, comments, or properties as desired.
- 3 To add or modify custom properties for the configuration, click **Custom**. For more information, see **Custom Properties** on page 5-21.
- 4 Click **OK**.

### **To delete a configuration:**

- 1 In the ConfigurationManager, click the name of the configuration you want to delete.
- 2 Press the **Delete** key (or click **Edit, Delete**), and click **Yes** to confirm.

---

**NOTE:** Deleting a configuration *does not* delete any features or components.

You cannot delete the *active* configuration.

---

## Opening a Document in a Named Configuration

---

When you open a model document that has more than one configuration, you can specify which configuration to display. Otherwise, the document is opened in the configuration in use when it was last saved.

You can also open an assembly using a specific configuration of its parts, where available.

### **To open a named configuration:**

- 1 Click **File, Open**.
- 2 Browse to the desired part or assembly document, select the **Configure** check box, and click **Open**.
- 3 In the **Configure Document** dialog box, click **Use Named Configuration** and select the desired configuration from the list.

- 4 To use a named configuration of the parts contained in an assembly, click **Use specified configuration name for all part references when available**, and enter the part configuration name. Any part that has a configuration with the specified name is opened in that configuration.

---

**NOTE:** The option in Step 4 does not affect any component instance whose component properties specify **Use Named Configuration**. See **Component Configurations in an Assembly** on page 8-7.

---

- 5 Click **OK**. The document opens in the selected configuration.

## Special Assembly Configurations

---

You can quickly create new assembly configurations, by either suppressing or resolving all the components at once. The easiest way to do this is to open the assembly in a new configuration, with one of these options:

- structure only shown (all components suppressed)
- all referenced models shown (all components resolved)

For more information about suppressed and resolved components, see **Suppression States** on page 6-40.

### *To suppress or resolve all components in a new assembly configuration:*

- 1 Click **File, Open**.
- 2 Browse to the desired assembly document, select the **Configure** check box, and click **Open**.
- 3 In the **Configure Document** dialog box, select either:
  - **New configuration showing all referenced models.** This opens the complete assembly. The components are all resolved and shown, regardless of the suppression and visibility state they were in when last saved. All components appear in the assembly model.
  - **New configuration showing assembly structure only.** This opens the assembly structure only. The components are all suppressed, regardless of the state they were in when last saved. No components appear in the assembly model until you change their suppression state to resolved.
- 4 Enter a new **Configuration name**, and click **OK**.

## Component Configurations in an Assembly

---

If a component (either an individual part or a sub-assembly) has more than one configuration, you can specify which configuration to use for each instance in the assembly.

### **To specify the configuration of a component instance:**

- 1 Right-click the component name in the FeatureManager design tree, or right-click on a component face, and select **Component Properties**.
- 2 Under **Referenced configuration**, select one of the following:
  - **Use component's "in-use" or last saved configuration**

Any instances with this option use the configuration that is currently active (if the document is open), or the configuration that was active when it was saved. When you open an assembly, any active components that are not already open in their own windows are loaded into memory. If you change the configuration in the component's own window, the newly selected configuration is placed in memory; any instances that use this option update when you return to the assembly.

---

**NOTE:** When you edit a part *in the context of an assembly*, the referenced configuration in the assembly becomes the active configuration in all open windows (for example, if the part is open in its own window, or in use in another open assembly, with this property selected).

---

To change the active configuration for a component that is not already open, right-click the component name in the FeatureManager design tree, and select **Open part.sldprt** or **assembly.sldasm**. Then make the desired configuration active.

You cannot change the referenced configuration of a suppressed component, because suppressed components are removed from memory. However, if you open the document of a suppressed component, and change the configuration, then the newly selected configuration will be used if you restore the component to the resolved state.

The component name in the FeatureManager design tree indicates whether a component uses the last saved or in-use configuration.

- **Use named configuration**

To *always* use a specific configuration, regardless of the configuration currently in memory, select this option, and choose a configuration from the list.

It is good design practice to use named configurations when components have multiple configurations, to prevent the in-use configuration from affecting the assembly in unexpected ways.

- **Use same configuration as pattern seed component**

If the selected component is an instance of a component pattern, this is the default setting. To specify a configuration that is different from the seed component, click **Use named configuration**, and choose a configuration from the list.

- 3 If the selected component is a component of a sub-assembly, choose which visibility properties to use:
  - **Use visibility properties specified in configuration <configuration\_name> of <sub-assembly\_name>.** The **Visibility** properties (**Hide Model**, **Color**) specified for this component in the selected configuration in the sub-assembly document are used.
  - **Override visibility properties specified in configuration <configuration\_name> of <sub-assembly\_name>.** The **Visibility** properties specified for this component in this configuration of the assembly are used.
- 4 Click **OK**.

## Design Tables

---

A design table allows you to build multiple configurations of parts or assemblies by specifying parameters in an *embedded* Microsoft Excel worksheet.

The design table is saved in the model document and is *not linked* to the original Excel file. Changes you make in the model are not reflected in the original Excel file.

To use design tables, it is strongly recommended that you have Microsoft Excel 97, with service pack SR-2, installed on your computer.

You can control the following items in a part design table:

- dimensions and suppression state of features
- configuration properties, including part number in a bill of materials, comments, and custom properties

You can control the following parameters in an assembly design table:

- components – suppression state, visibility, referenced configuration
- assembly features – dimensions, suppression state
- mates – dimensions of distance and angle mates, suppression state
- configuration properties – part number and display in a bill of materials (when used as a sub-assembly), comments, custom properties

## Creating a Design Table

To create a design table, you define the names of the configurations you want to create, you specify the parameters you want to control, and you assign values for each parameter.

There are two ways to create a design table:

- Create a design table worksheet as a separate operation in Excel. Save the worksheet, then insert it in the model document to create the configurations.
- Insert a new, empty design table in the model, and enter the design table information directly in the worksheet. When you are done entering the design table information, the new configurations are automatically created in the model.

With this method, it is easier to add some types of parameters to the worksheet. See page 8-13 through page 8-14 for more information.

Either way, you can insert a partially completed worksheet, then edit it later to add more configurations, to control additional parameters, or to update values.

---

**NOTE:** By default, you edit a design table in the SolidWorks window, as an embedded object.

If you prefer to work in a separate Excel window, click **Tools, Options, General**, and select **Edit design tables in separate window**.

---

**To create a design table as a separate Excel file:**

- 1 Open Microsoft Excel and create a worksheet.
- 2 In the first *column* (row header cells **A2, A3**, etc.), enter the names of the configurations that you want to create. Leave cell **A1** *blank*.

The names must not include the forward slash (/) or at (@) characters.

---

**NOTE:** If a configuration name contains only numbers, Excel interprets the name as a numeric value, instead of text. This is not a problem, except in the following cases:

- If the value begins with zeroes, the leading zeroes are removed.
- If the value is more than 12 characters long, it is displayed in scientific notation.

To correct the problem, right-click the header cell for row **A** (to select the whole column), and select **Format Cells**. On the **Number** tab, under **Category**, select **Text**.

---

- 3 In the first *row*, (column header cells **B1, C1**, etc.), enter the parameters that you want to control.

	A	B	C
1		parameter1	parameter2
2	configuration1	value1	value3
3	configuration2	value2	value4


See **Specifying Design Table Parameters** on page 8-12 for detailed information.

- 4 Enter the parameter values in the spreadsheet cells.
- 5 Save the worksheet.
- 6 Open the model document where you want to insert the design table.
- 7 Click **Insert, Design Table**. In the **Open** dialog box, browse to the Excel worksheet (.xls) and click **Open**.

The worksheet appears in the model document, and the Excel menus and toolbars replace the SolidWorks menus and toolbars. (If you selected the **Edit design tables in separate window** option, the design table is opened in an Excel window.)

A message is displayed, indicating the names of any new configurations that were generated by the design table.

- 8 Edit the design table, if necessary. When you are finished editing, click anywhere outside the table to close it. The SolidWorks menus and toolbars reappear.

To display the configurations added by the design table, click the Configuration tab  at the bottom of the window. Double-click the name of the configuration, or right-click the name and select **Show Configuration**.

### **To insert a new design table:**

- 1 Click **Insert, New Design Table**.

An embedded worksheet appears in the window, and the SolidWorks toolbars are replaced with Excel toolbars. (If you selected **Edit design tables in separate window** under **Tools, Options, General**, the worksheet opens in a separate Excel window.)

Cell **A1** identifies the worksheet as **Design Table for: *model\_name***. Cell **A3** contains the default name for the first new configuration, **First Instance**.

- 2 In row **2**, enter the parameters that you want to control. (Leave cell **A2** blank.) Notice that cell **B2** is active.

---

**TIP:** There is an easy way to add certain types of parameters in a new design table. With the appropriate worksheet cell active:

- **Dimensions.** Double-click a dimension in the graphics area. (Make sure that the necessary dimensions are displayed *before* you insert the new design table.)  
The ***Dimension@Feature*** or ***Dimension@Sketch*** parameter is inserted in the cell.
- **Feature suppression.** Double-click a face of the feature.  
The ***\$STATE@Feature*** parameter is inserted in the cell.
- **Component suppression.** With the appropriate worksheet cell active, double-click a face of the component.  
The ***\$STATE@component<instance>*** parameter is inserted in the cell.

As you continue to add parameters this way, the adjacent cell (**C2**, **D2**, and so on) is activated automatically. Each parameter is added to the header row, and the current value is displayed in row 3 (**First Instance**).


---

- 3 In column **A** (cells **A3**, **A4**, etc.), enter the names of the configurations that you want to create. The names must not include the forward slash (/) or *at* (@) characters. See the **NOTE** on page 8-9.

You can change the configuration name in cell **A3** (**First Instance**) if you want.

- 4 Enter the parameter values in the spreadsheet cells. See **Specifying Design Table Parameters** on page 8-12 for detailed information.

	A	B	C
1	<b>Design Table for: Part3</b>		
2		<b>parameter1</b>	<b>parameter2</b>
3	<b>First Instance</b>	<i>value1</i>	<i>value4</i>
4	<b>configuration1</b>	<i>value2</i>	<i>value5</i>
5	<b>configuration2</b>	<i>value3</i>	<i>value6</i>

- 5 When you are done adding information to the worksheet, click outside the table to close it. (If you are working with the design table in a separate window, click **File, Close & Return to *model\_name***.)  
A message is displayed, listing the configurations that were created.
- 6 To display the configurations, click the Configuration tab  at the bottom of the window. Double-click the name of the configuration, or right-click the name and select **Show Configuration**.

**To edit a design table:**

- 1 Click **Edit, Design Table**.

The worksheet appears in the window. (If you selected **Edit design tables in separate window** under **Tools, Options, General**, the worksheet is opened in a separate Excel window.)

- 2 Edit the table as needed. You can change the parameter values in the cells, add rows for additional configurations, or add columns to control additional parameters.

You can also change the appearance of the design table by editing the format of the cells. Use the Excel functions to modify fonts, alignment, borders, and so on.

- 3 Click outside of the table to close it. (If you are working with the design table in a separate window, click **File, Close & Return to *model\_name***.)

The configurations update to reflect your changes.

---

**NOTE:** Changes you make directly in the model (not in the table) are not reflected in the original Excel worksheet, or in the embedded table. The next time that the design table is evaluated, the configurations are updated to reflect the data in the table.

For example, assume that **D1@Sketch1** is driven by the design table. If you modify **D1@Sketch1** outside of the table (by editing the sketch, when one of the configurations created by the table is active), *the table data does not change*. Furthermore, the value contained in the corresponding table cell is restored to the configuration the next time that the table is evaluated, and *your change is undone*.

---



**To delete a design table:**

- 1 Click **Edit, Delete Design Table**.
- 2 Confirm the deletion and click **OK**.

Deleting the design table *does not* delete the configurations created by it.

**To delete a configuration created by the design table:**

- 1 Activate a configuration that you want to keep. The configuration that you want to delete must not be active.
- 2 Click **Edit, Design Table**.
- 3 Click the numbered cell beside the configuration name that you want to delete (this selects the whole row), then click **Edit, Delete**.

You can also right-click the numbered cell, and select **Delete** from the shortcut menu.

**IMPORTANT:** Do not press the **Delete** key to delete the row. Pressing **Delete** clears the contents of the cells, leaving an *empty* row. Any rows below an empty row are *ignored*.

When you click outside the table, the table is evaluated, and you are asked to confirm the deletion of the configurations.

---

**NOTE:** You must use this method to permanently delete any configuration that is created by the design table. Deleting design table configurations from the ConfigurationManager is not sufficient.

All configurations in the design table are evaluated when you edit and close the table. Therefore, if you delete a configuration from the ConfigurationManager (without deleting it from the table), then edit the design table in any other way, the configuration is restored.

---

## Specifying Design Table Parameters

When used in a design table, the names of dimensions, features, components, and configurations must match the name in the model. To ensure an exact match, you can copy and paste the name from the **Properties** dialog box for the selected item.

**To copy and paste a feature or dimension name:**

- 1 Right-click the item (dimension, feature, or configuration), and select **Properties**. For a component, select **Component Properties** from the shortcut menu.
- 2 In the **Properties** dialog box, copy (**Ctrl+C**) the item name (for dimensions and components, be sure to copy the **Full name**).
- 3 Paste (**Ctrl+V**) the name into the appropriate column header cell.

Some parameter specifications that appear in the column header cells include a *keyword*, such as **\$CONFIGURATION**, or **\$COMMENT**. The parameter keywords are *not* case-sensitive.

Some parameters (suppression state, visibility, or referenced configuration of a component) also include a component name and instance number. These parameters use this syntax:

***\$KEYWORD@component<instance>***

where ***component*** is the name of the component, and ***instance*** is the instance number (an integer), in angle brackets.

For example, the column labeled ***\$CONFIGURATION@gear<2>*** controls the referenced configuration of the second instance of the component named ***gear***.

You can specify multiple instances of a component in a single column header cell. Use the following syntax for the instance numbers:

- <\*>*** all instances
- <1-4>*** range of instances
- <1,3,6>*** list of non-consecutive instances, separated by commas
- <1,3-6,8>*** combination, separated by commas

Values that are not item names (***Yes***, ***Suppress***, and so on.) are *not* case-sensitive. The syntax and legal values for each parameter type are described below.

This is an example of a design table that creates three configurations, adds a comment for each one, suppresses a feature, and controls a dimension.

	A	B	C	D
1		<b><i>\$COMMENT</i></b>	<b><i>\$STATE@Cut-Extrude1</i></b>	<b><i>D3@Sketch2</i></b>
2	<b><i>rev1</i></b>	large, no cuts	suppressed	125
3	<b><i>rev2</i></b>	medium	unsuppressed	100
4	<b><i>rev3</i></b>	small	unsuppressed	75

---

**NOTE:** Do not include any rows or columns with blank headers in the middle of the design table. The software stops evaluating the data when it reaches a row or column that has an empty header cell. Data that appears in or after a row or column with an empty header is ignored.

---

### ***Dimensions***

In a part document, you can use a design table to control the dimensions in sketches and in feature definitions.

In an assembly document, you can control dimensions that belong to assembly features. This includes mates (angle, distance), assembly feature cuts and holes, and component patterns (spacing, instance count). You cannot control the dimensions of a component model contained in the assembly.

A dimension name is in the form:

***Dimension@Feature*** or ***Dimension@Sketch<n>***

For example, the full name for the depth of an extruded base feature is **D1@Base-Extrude**; the full name for the dimension of the first **Distance** mate is **D1@Distance1**. You can use the **Properties** command to assign meaningful names to dimensions. See **Dimension Properties** on page 10-12.

If you are inserting a new, empty design table, see the **TIP** on page 8-10.

In the table body cells, enter the value for the dimension. If a cell is left blank, the default value is zero; if zero is not a legal value for this dimension, an error message is displayed.

---

**NOTE:** When you specify values, be sure to use the system of units specified for the model document (under **Tools, Options, Grid/Units**).

---

### ***Component configuration***

The column header for controlling component configuration uses this syntax:

***\$CONFIGURATION@component<instance>***

In the table body cells, enter the name of the desired configuration. Blank cells are not allowed in this column.

### ***Visibility***

The column header for controlling component visibility uses this syntax:

***\$SHOW@component<instance>***

In the table body cells, to show the model, enter **Yes** (or **Y**). To hide the model, enter **No** (or **N**). If a cell is left blank, the default is **No** (do not show the model).

### ***Component suppression state***

The column header for controlling the suppression state of a component uses this syntax:

***\$STATE@component<instance>***

In the table body cells, enter the value for the desired suppression state: **Suppressed** (or **S**), or **Resolved** (or **R**). You cannot set a component to the lightweight state. If a cell is left blank, the default suppression state is **Resolved**.

If you are inserting a new, empty design table, see the **TIP** on page 8-10.

### ***Suppression of features***

In a part document, you can suppress any feature.

In an assembly document, you can suppress features that belong to the assembly. These include mates, assembly feature holes and cuts, and component patterns. Sketches and reference geometry may also belong to an assembly. You cannot control the suppression of a feature that belongs to an individual assembly component.

There are two ways to specify the suppression of features.

- ❑ **Method 1 (recommended).** The column header for controlling feature suppression uses this syntax:

**\$STATE@feature**

where *feature* is the name of the feature. For example, the column labeled **\$STATE@Hole1** controls the suppression of the first hole.

In the table body cells, enter the value for the desired suppression: **Suppressed** (or **S**), **Unsuppressed** (or **U**). If a cell is left blank, the default is **Unsuppressed**.

If you are inserting a new, empty design table, see the **TIP** on page 8-10.

- ❑ **Method 2.** Enter only the feature name in column header cell. To suppress the feature, leave the table body cell *blank*. To include the feature, enter any string in the body cell. This is the syntax that was used in SolidWorks 98 and earlier versions, and is included for backward compatibility.

### **Part number**

When you create configurations with a design table, the software automatically sets options in the **Configuration Properties, Advanced** dialog box. The **Use name below** option is selected, and the configuration name is used. A bill of materials for an assembly containing the document lists the configuration name in the **Part No.** column.

The **\$PARTNUMBER** parameter lets you specify a *different* value for use in the **Part No.** column of the BOM. For an assembly document, this is the part number used when this configuration is used as a sub-assembly. The column header uses this syntax:

**\$PARTNUMBER**

In the table body cells, enter the part number for each configuration. If a cell is left blank, the default is to use the configuration name.

If multiple configurations of the same document are used in an assembly, the BOM lists each configuration by name as a separate item number. If you *do not want* each configuration listed separately in the BOM, do one of the following:

- Assign the same value for the **\$PARTNUMBER** parameter in all the configurations.
- After the design table creates all the configurations, edit the advanced properties of each configuration, and select **Use document name**. To override the document name, you can use the **Title** specified under **File, Properties** for the component document. For more information, see **Bill of Materials (BOM)** on page 10-31.

### **Comment**

The **Configuration Properties** dialog box has a **Comment** box, where you can enter a description, or other additional information about the configuration. The column header for entering information in this box for each configuration uses this syntax:

**\$COMMENT**

In the table body cells, enter the comments for the configuration. Comments are optional; if a cell is left blank, the **Comment** box is empty.

### ***Expand in BOM***

You can control how the assembly is listed in a BOM, when this configuration is used as a sub-assembly. The column header uses this syntax:

**\$NEVER\_EXPAND\_IN\_BOM**

This corresponds to the **Don't show child components in BOM when used as a sub-assembly** option in the **Configuration Properties, Advanced** dialog box.

- When turned *off*, the sub-assembly is expanded if you select either **Show parts only** or **Show assemblies and parts in an indented list** when you insert the BOM.
- When turned *on*, the sub-assembly is *never* expanded, regardless of the option you select when you insert the BOM. The effect is the same as selecting **Show top-level subassemblies and parts only**.

In the table body cells, to turn on the option (never expand) enter **Yes** (or **Y**). To turn off the option (allow to expand), enter **No** (or **N**). If a cell is left blank, the default is **No**.

For more information, see **Bill of Materials (BOM)** on page 10-31.

### ***Custom properties***

The column header for specifying a custom property uses this syntax:

**\$PRP@*property***

where ***property*** is the name of a custom property. You can use one of the custom properties listed in the **Summary Info** dialog box (click **File, Properties, Configuration specific**), or you can add a new custom property.

---

**NOTE:** If you plan to use the custom properties as columns in the bill of materials, do not use property names that contain spaces. See **Custom properties in a bill of materials** on page 10-36.

---

For example, use the header **\$prp@Cost** to control the custom property **Cost** for each configuration. In the table body cells, enter the property value for each configuration. If a cell is left blank, the property is undefined for the configuration.

### ***User notes***

The design table can include additional columns or rows for information only (notes, intermediate calculations, etc.). Use the keyword **\$USER\_NOTES** in the header of any column or row that you do not want evaluated. You may use as many of these rows or columns as you need. However, you may not include rows or columns with *empty* headers in the middle of the table.

## Summary of Design Table Parameters

Table 8-1. Summary of design table parameters

parameter syntax (header cell)	legal values (body cell)	default if value left blank
<b>parts and assemblies</b>		
\$comment	any text string	empty
\$partnumber	any text string	configuration name
feature_name	blank (suppressed) any string (unsuppressed)	Suppressed
\$state@ feature_name	Suppressed, S Unsuppressed, U	Unsuppressed
dimension@feature	any legal decimal value for the dimension	zero (error if not a legal value)
\$prp@property	any text string	property is undefined
\$user_notes	any text string	not evaluated
<b>assemblies only</b>		
\$show@ component<instance>	Yes, Y No, N	No
\$state@ component<instance>	Resolved, R Suppressed, S	Resolved
\$configuration@ component<instance>	configuration name	not allowed
\$never_expand_in_BOM	Yes (never expand) No (allow to expand)	No

## Design Tables in Drawings

If a model document uses a design table to generate multiple configurations, you can display the table in a drawing of that model. That way, a single drawing can represent all the configurations.

You can label the dimensions with letters or short, descriptive names, instead of using the full dimension names and the values. For example, instead of **D1@BaseExtrude**, you can use a label such as **A** or **thickness** as the column header in the table, and display the label instead of the value of the dimension in the drawing view.

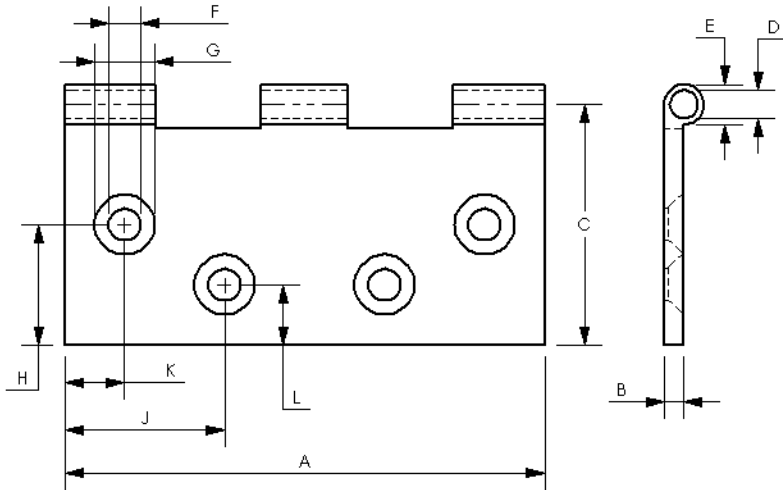
### *To display a design table in a drawing:*

- 1 In the part or assembly document, edit the design table as needed. The design table is displayed in the drawing exactly as it appears in the model document.
  - Drag the resize handles on the corners and sides of the table to adjust the boundaries of the table. Make sure that all the rows and columns that you want are displayed, and remove any empty rows or columns.
  - Hide any rows (or columns) in the middle of the table that you do not want to display in the drawing. Right-click the numbered cell at the left of the row (or the lettered cell at the top of the column) to select the whole row (or column), and select **Hide**.
  - Adjust the column width, row height, text alignment, borders, fonts, and so on, as needed to meet your company standards.
- 2 If you want to use labels (letters or names) for the dimensions:
  - a) Insert a new row *between* the header row (the row containing the full dimension names) and the row for the first configuration.
  - b) In the new row, enter a label for each dimension.
  - c) Hide the header row as described in Step 1.
- 3 In a drawing of the part or assembly, select a drawing view.
- 4 Click **Insert, Design Table**. The design table appears, and you can drag it into place on the sheet.
- 5 If you need to change the size of the design table, right-click the table, and select **Properties**. Enter a **Scale** value, or specify a value for **Height** or **Width**. When you change any one of the values, the others update automatically, so that the proportions of the table are maintained.

To restore the table to its original size, right-click the table, and select **Reset Size**.
- 6 If you are using labels, modify the dimensions in the drawing view:
  - a) Right-click a dimension, and select **Properties**.
  - b) Click **Modify Text**.
  - c) Delete **<DIM>** (the placeholder for displaying the dimension value) from the middle line, then enter the corresponding label from the table.

For more information about dimensions in drawing views, see **Importing Model Dimensions** on page 10-4 and **Modify Text** on page 10-16.

The following illustration is an example of how a design table and the dimensions it controls might appear in a drawing.



	A	B	C	D	E	F	G	H	J	K	L
001-080	80	4	50	5	8	5	9	25	26	10	10
001-120	120	5	60	7	10	8	15	30	40	15	15
001-200	200	8	90	13	16	8	15	45	66	20	15

**To edit the design table:**

- 1 Double-click the design table in the drawing. The model is opened in its own window, and the design table is opened for editing.

The table appears either embedded in the SolidWorks window, or in a separate window if you selected the **Edit design table in separate window** option. (See **General Options** on page A-13.)

- 2 Modify the table as needed. When you are done, click outside of the table to close it.

If you are working with the design table in a separate window, click **File, Close & Return to model\_name**.

When you return to the drawing window, the table updates.



# Drawings

---

You can create 2D drawings of the 3D solid parts and assemblies you design. Parts, assemblies, and drawings are linked documents; any changes that you make to the part or assembly change the drawing document.

---

**NOTE:** One-way associativity can be set between drawings and models during the software installation. This prevents changes to model dimensions, thereby preventing changes to the model itself from within a drawing. This option can only be reset with a new installation of the software.

---

Generally, a drawing consists of several views generated from the model. Views can also be created from existing views. For example, a section view is created from an existing drawing view.

This chapter describes:

- Setting default drawing and page set-up options
- Working with drawing templates and sheets
- Creating drawings with standard views
- Creating other views, such as detail, section, projected, broken, etc.
- Aligning and displaying views
- 2D sketching

Dimensions, notes, datums, geometric tolerances, and other notations complete the drawing. Dimensions and other annotations are discussed in Chapter 10, “Detailing.”

## Drawing and Page Setup Options

---

You have a variety of options to customize your drawings to your company's standards and to conform to the requirements of your printer or plotter.

To customize your SolidWorks drawings, you can:

- Select default drawing options and behaviors by clicking **Tools, Options, Drawings**. Specify the scale, and various display and update options for views. See **Drawings Options** on page A-8.
- Select document-specific drawing options by clicking **Tools, Options**, and then reviewing the options available on these tabs:
  - **Crosshatch**. Specify the crosshatch patterns used for section views, and for user-defined hatched areas. See **Crosshatch Options** on page A-3.

---

**NOTE:** The crosshatch pattern specified in the *part* documents is used for section views created for the part (and for instances of the part in assemblies). See **Modifying the crosshatch** on page 9-23.

---

- **Line Font**. Specify the line thicknesses and styles used for various edge types in the drawing. See **Line Font Options** on page A-17.
- **Detailing**. Specify numerous options related to dimensions and other types of annotations. See **Detailing Options** on page A-4.
- Set the page properties to meet your printer or plotter requirements. Click **File, Page Setup** to specify:
  - Paper margins and orientation
  - Line weights
  - Scale of printed drawing
  - Custom headers and footers
  - Printing drawings in colorFor more information, see **Page Setup** on page 1-33.
- Customize your drawing templates to conform to the standards of your workplace. You can:
  - Customize information blocks and text (See **Customizing a Drawing Template** on page 9-5.)
  - Change the sheet size and orientation (See **Modifying the Sheet Setup** on page 9-3.)


## Creating a Drawing

---

Drawings consist of one or more views generated from a part or assembly. The part or assembly associated with the drawing must be saved before you can create the drawing.

Drawing files have the **.slddrw** extension.

### **To create a new drawing:**

- 1 Click **New**  on the Standard toolbar, or click **File, New**.
- 2 In the **New** dialog box, select **Drawing** and click **OK**.
- 3 In the **Template To Use** dialog box, select a template:
  - **Standard Template.** Select a standard sheet size template (for example, **A-Landscape**) from the list.
  - **Custom Template.** Click **Browse**, navigate to a custom template on your system or the network, and click **Open**. See **Customizing a Drawing Template** on page 9-5 for information about creating a custom template.
  - **No Template.** Select a blank, standard sheet from the **Paper Size** list, or select the **User Defined** option. If you select **User Defined**, you must specify the paper size (**Width** and **Height**).
- 4 Click **OK**.

A new drawing document is opened, using the selected template, and the current sheet scale is displayed in the status bar at the bottom of the SolidWorks window.

## Modifying the Sheet Setup

You can set up the sheet details when you start a drawing, or later. You can also modify existing sheet details. If you choose **No Template** when you open a new drawing, the default values specified in the **Drawings Options** (see page A-8) are used.

### **To specify sheet details:**

- 1 Right-click the sheet icon in the FeatureManager design tree, or any blank area of the drawing sheet, or the sheet tab at the bottom of the drawing window, and select **Properties**.

You can only edit the properties of the *active* sheet. For information about creating and activating additional sheets, see the next section.
- 2 Make the appropriate entries on the **Sheet Setup** dialog box:
  - **Name.** Enter a title in the box.
  - **Paper size.** Select a standard size from the list, or select **User Defined** to specify a custom paper size. If you selected **User Defined**, specify a **Height** and **Width** for the paper.
  - **Scale.** Specify the default scale for all views on the sheet.

- **Template.** Select a standard template from the list, or select **Custom** or **None**. If you select **Custom**, use the **Browse** button to select a custom template.
- **Type of projection.** Select **First angle** or the default, **Third angle**.
- **Next view label, Next datum label.** Specify the letter of the alphabet that will be used.
- **Use custom property values from model shown in.** This option is used only if more than one model document is shown on the sheet. If the drawing has notes that are linked to custom file properties of a model, select which view contains the model whose properties you want to use. If you do not specify otherwise, the properties of the model in the *first* view added to the sheet are used. See **Linking Notes to Document Properties** on page 9-7.

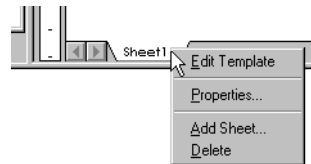
3 Click **OK**.

## Multiple Sheets

You can add sheets to the drawing at any time.

### To add a sheet:

- 1 Click **Insert, Sheet**. You can also right-click any sheet tab or any sheet icon in the FeatureManager design tree, and select **Add Sheet**.



The **Sheet Setup** dialog box appears, with the default name of the next new sheet in sequence, for example, **Sheet2**.

- 2 Specify the sheet details as described in the previous section, and click **OK**.

### To view a different sheet:

Click the sheet tab of the desired sheet. You can also right-click the sheet tab or sheet icon in the FeatureManager design tree, and select **Activate**.

### To delete a sheet:

- 1 Right-click any sheet tab or any sheet icon in the FeatureManager design tree, and select **Delete**. You can also click anywhere in the graphics area and press the **Delete** key to delete the active sheet.
- 2 Click **Yes** in the **Confirm Delete** dialog box.

## Templates, Sheets, and Views

Ownership of items in drawings is dependent on what is active when the item is created. Sketch entities, notes, and other items can belong to the drawing sheet, to an individual view, or to the template. In general, you must activate the owner of an item in order to edit the item.

---

**NOTE:** Dimensions and annotations that are attached to model geometry are exceptions to this rule. They belong to the view where they are attached, regardless of whether the view is active when you create the item.

---

For example, you can activate a view, and create a construction line that belongs to the view. If you move or delete the view, the construction line is moved or deleted also. To edit the construction line, you must activate the view again.

You could create the same construction line on the sheet, when no view is active. However, if you move or delete the views on the sheet, the construction line is not affected. To edit the construction line, the sheet itself must be active.

Similarly, you must activate the template to edit any items that are owned by the template. While the template is active, the sheet is hidden, including all the items that belong to the sheet, and all the views.

## Customizing a Drawing Template

You can customize drawing templates to match your company's standard format.

**To edit the drawing template:**

**1 Click Edit, Template.**

You can also right-click any blank area on the drawing sheet, or the sheet icon in the FeatureManager design tree, and select **Edit Template**.

If the sheet does not already have a template, you create one if you select **Edit Template**, then click **Rebuild**.





If the sheet already has a template, you can also right-click the template icon in the FeatureManager design tree, and select **Edit Template**.

**2 To edit existing text in the title blocks, double-click the text.**

The **Properties** dialog box for **Note** appears.

- Enter new text in the **Note text** box.
- Modify the **Angle** and **Text justification (Center, Left, Right)** if needed.

ITEM NO.	PART OR IDENTIFYING NO.	NUMBER/QUANTITY OR DESCRIPTION	MATERIAL SPECIFICATION	QTY REQD
PARTS LIST				
UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE		CAD GENERATED DRAWING. DO NOT MANUALLY UPDATE		
FRACTIONS DECIMAL ANGLES		INSERT COMPANY NAME HERE		
+	.XXX .XX + 1	DATE		
-	.XXX .XXX	CHECKED		
MATERIAL	--	APP'D		
FINISH	--	INFO ENG		
D.O. NOT SCALE DRAWING	SCALE	SCALE	DATE	SHEET 1 OF 1

- To use a different **Font**, clear the **Use document's font** check box, then click **Font** to select a new font style or size.
  - To specify a link to a file or URL, click **Insert Hyperlink** .
  - To link the text of the note to a property, click **Link to Property** . See **Linking Notes to Document Properties** on page 9-7.
- 3 Click **OK** to accept the new font and/or text.
  - 4 To edit the template format by moving, deleting, and adding lines or text in the text blocks:
    - To delete, click the line or text and press the **Delete** key.
    - To move, click the line or text and drag to a new location.
    - To add lines, click **Line**  on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Line**.
    - To add text, click the sheet, then click **Note**  on the Annotations toolbar, or click **Insert, Annotations, Note**. Specify the text properties, then drag the text to the desired location.
  - 5 The standard templates supplied with the software include an anchor point for placing the bill of materials in a specific location on the sheet. To set the anchor point, expand the **Template** icon for the sheet. Right-click the **Bill of Materials Anchor** icon, select **Set anchor**, then click where you want the anchor point on the sheet.

You can also right-click a sketch point in the template, and select **Set as Anchor**. The anchor point is highlighted in pink, and its visibility toggles when you click the icon.

---

**NOTE:** You can add an anchor point to drawings that you created in versions earlier than SolidWorks 98Plus.

Right-click the sheet, and select **Edit Template**. Then right-click a sketch point (add one if needed), and select **Set as Anchor**.

If the sheet does not already have a template, you create one when you perform this procedure.

---

For information about using the anchor point when you insert a BOM, see **Bill of Materials (BOM)** on page 10-31.

- 6 To add objects from other applications, such as bitmaps of your company logo, standard note text from a file, and so on, click **Insert, Object**.

You can either create a new object or insert one from a file. Objects may be *linked* or *embedded* in template files. For more information, see the section about object linking and embedding (OLE) in Chapter 1, “SolidWorks Fundamentals.”

### **To save the template:**

Click **File, Save Template**. The default location where templates are stored is `<install_directory>\data`, unless you specified a different **data** folder when you installed the software.

- To overwrite a **Standard Template**, select the template from the list, and click **OK**.
- To save the template with a new name, click **Custom Template**. Click **Browse**, navigate to the desired directory, enter a new template name, and click **Save**. (Template files have the **.slddrt** extension.)

When you save the template, any custom file properties that you have added to the drawing document are saved with the template. The custom properties in the template are added to any new drawing that uses the template.

See **Custom Properties** on page 5-21 for more information.

### **To end editing the template and return to editing the sheet:**

Click **Edit, Sheet**. You can also right-click the sheet icon or template icon in the FeatureManager design tree, or right-click any blank area of the sheet, and select **Edit Sheet**.

## **Linking Notes to Document Properties**

To automatically insert information in a drawing, you can link note text in the template to document properties.


All SolidWorks documents have the system-defined property **SW-File Name** (the name of the document, without the extension). Additionally, drawings have the following system-defined properties:

<b>Property name</b>	<b>Value</b>
<b>SW-Sheet Scale</b>	scale of the active sheet
<b>SW-Template Size</b>	sheet size of the active sheet
<b>SW-Current Sheet</b>	sheet number of the active sheet
<b>SW-Total Sheets</b>	total number of sheets in the active drawing document
<b>SW-Sheet Name</b>	name of the active sheet

You can link a note to the properties of the model that is shown in the drawing (the **SW-File Name** property, or a user-defined custom property in the model document).

While you are editing the template, a variable for the property name is displayed (**\$PRP: "property name"**). When you return to editing the sheet, the value of the property is displayed, if it is found. If the property value cannot be found, the variable for the property name is displayed on the sheet.

**To link a note in the template to a property:**

- 1 In the **Properties** dialog box for the **Note**, click **Link to Property** .
- 2 In the **Link to Property** dialog box, select a **Property Name** from the list. The list contains the names of the system-defined properties, and any custom properties to which you have assigned values in *the active drawing document*.

A linked note may include additional text, and it may include links to more than one property. For example, to display the current sheet number and the total number of sheets, you can add this note:

**SHEET \$PRP: “SW-Current Sheet” OF \$PRP: “SW-Total Sheets”**

On the sheet, the property values are displayed:


**SHEET 1 OF 2** (on the first sheet of a two-sheet drawing)

- 3 To link the note to the properties of *the model shown in the drawing*, click **External model reference**, then select a **Property Name** from the list. The value of the property for the model shown in *the first view* inserted into the sheet is displayed by default.

If the sheet contains views of more than one model, you can specify which view contains the model whose properties you want to use. Right-click the sheet, select **Properties**, then select the view from the list under **Use custom property values from model shown in**.

If no model is shown in the drawing (for example, when you are creating a new template), you may either select a property from the drawing’s list, or add a new custom property. The property is resolved when a model is added to the sheet.

When you save the template, any new properties that you added are saved with the template. The properties are added to any new drawing that uses the template.

- 4 Click **Rebuild**  to update the drawing, if necessary.



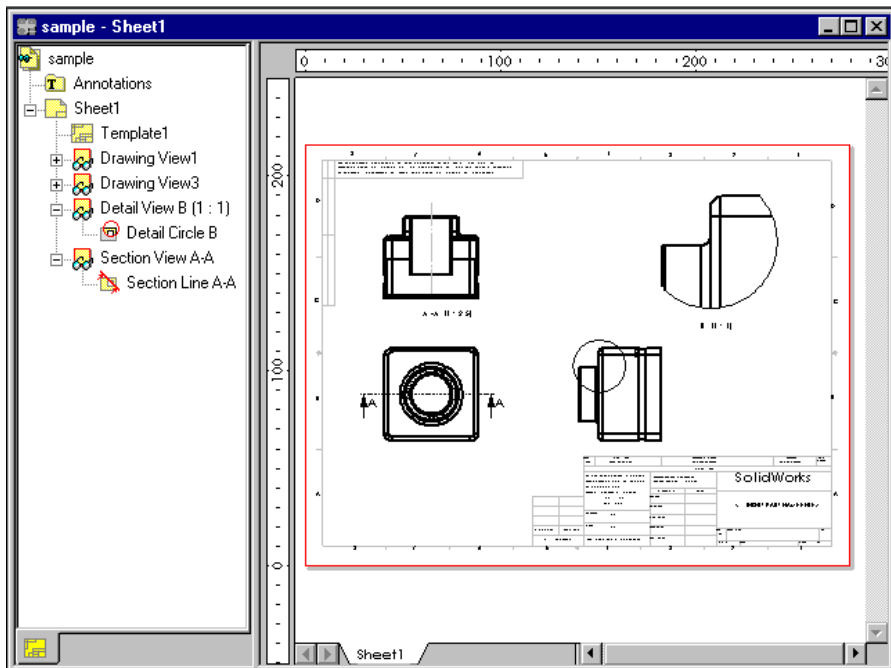
## The Drawing Window

The drawing window includes a FeatureManager design tree that is similar to the design tree in the part and assembly windows. The FeatureManager design tree for drawings consists of a hierarchical list of items pertaining to the drawing. There is an icon for each sheet; under each sheet, there are icons for the template and each view.

A **+** symbol beside an item's icon indicates that it contains associated items. Click the **+** to expand the item and display its contents.

Standard views contain the feature list for the part or assembly shown in the view. Generated views, such as detail or section views, contain different, view-specific items (detail circles, section lines, and so on).

The drawing window has rulers at the top and left side. The rulers (and the status line) report the position of the pointer on the sheet. To turn the ruler display on or off, click **View, Rulers**.



## Views of Parts and Assemblies

---

You can generate drawings of a part or assembly in a variety of view types: **Standard 3 View**, **Named**, and **Relative to Model** views. (The part or assembly document must be saved before you can create drawing views of it.)

You can create **Projection**, **Auxiliary**, **Detail**, **Section**, **Aligned Section**, or **Broken** views from existing views on the active sheet.

You can also create an **Empty** view for sketching. See **2D Sketching** on page 9-39 for more information.

For some operations, you need to *select* a view; for others, you *activate* a view.

- ❑ *Selecting* a view allows you to create a projected view, to insert break lines for a broken view, to move the view on the sheet, and to resize the view boundary. A selected view has a green border with drag handles. To select a view, click an empty area within the view boundary.
- ❑ *Activating* a view allows you to sketch entities on that view. This is necessary for sketching section lines, and for defining areas for detail views. To edit the sketch entity, the view to which it belongs must be active. The border of an active view is a shadowed box.

You can set an option that causes drawing views to activate automatically when the pointer passes over them. Click **Tools, Options, Drawings**, then select or clear the **Dynamic drawing view activation** check box.

- ❑ When **Dynamic drawing view activation** is *selected*, the view *closest to the pointer position* is activated automatically. To stop the dynamic activation temporarily, you can lock the focus on a view or on the sheet. Right-click the view or the sheet, and select **Lock View Focus** or **Lock Sheet Focus**.
  - **Lock View Focus** allows you to add sketch entities to a view, even when the pointer is close to another view. You can be sure that the items you are adding belong to the view you want.
  - **Lock Sheet Focus** allows you to add sketch entities to the *sheet*. Otherwise, the sketch entities belong to the view that is closest to where you begin sketching.


To return to dynamic activation mode, right-click again, and select **Unlock View Focus** (or double-click a different view) or **Unlock Sheet Focus** (or double-click any view).

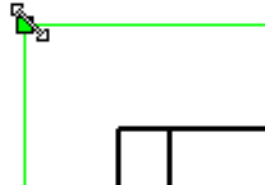
- ❑ When **Dynamic drawing view activation** is *cleared*, double-click anywhere within a view's boundary to activate the view, or right-click within the boundary and select **Activate View**. To de-activate a view, either double-click again, or activate a different view, or right-click and select **Activate Sheet**.

## View Boundaries

When the pointer passes over the edge of a drawing view, the view boundary is highlighted. The size of the view boundary is calculated automatically, based on the size, shape, and orientation of the model shown in the view. You can adjust the size of the view boundary (enlarging the view boundary may make it easier to select or activate a view). You cannot make the view boundary smaller than the model displayed in the view. View boundaries, and the views they contain, can overlap.

### ***To change the size of the view boundary:***

- 1 Select the view.
- 2 Point to a green drag handle at the corner or side of the boundary.
- 3 When the pointer displays a resize  shape, drag the handle as needed to resize the boundary.



### ***To turn the boundary display on/off:***

Click **Tools, Options, Drawings**, then select the **Display drawing view borders** check box. When selected, the boundary of each view is displayed as a thin gray box. When cleared, the boundaries are hidden. This option applies to *all* the views in the drawing.

Even if the boundaries are hidden, they still are highlighted when the pointer passes over them, and an active view still displays a shadowed box.

If the model contained in the view is not in the active or last-saved configuration, the border is dashed.



## Standard 3 View



The **Standard 3 View** option under **Insert, Drawing View** creates three default orthographic views of a part or assembly displayed at the same time.


There are several ways to create a Standard 3 View drawing.

### **To create the Standard 3 View:**

- 1 Open the part or assembly document, or a drawing document that contains a view of the model you want.
- 2 Open a new drawing.
- 3 Click **Standard 3 View**  on the Drawing toolbar, or click **Insert, Drawing View, Standard 3 View**. The pointer changes to .
- 4 Select the model in one of these ways:
  - To add the views of a *part*, in a part window, click a face, or anywhere in the graphics area, or click the part name in the FeatureManager design tree.
  - To add the views of an *assembly*, in an assembly window, click an *empty* region of the graphics area, or click the assembly name in the FeatureManager design tree.
  - To add the views of an *assembly component*, in an assembly window, click a face on the part, or click the name of either the individual part or the sub-assembly in the FeatureManager design tree.
  - In a drawing window, click a view that contains the desired part or assembly, either in the FeatureManager design tree or on the sheet.

The drawing window returns to the front (if necessary) and the three views are placed in the drawing.

If you prefer not to open the model document, you can still create a drawing of it.

- 1 In a drawing, click **Standard 3 View** .
- 2 Right-click in the graphics area and select **Insert From File**.
- 3 Browse to the desired model document, and click **Open**.

### **To create the Standard 3 View (drag-and-drop method):**

The default view created when you drag and drop a part or assembly into a drawing is the **Standard 3 View**.

- 1 Open a new drawing window.
- 2 Drag a part or assembly document from Windows Explorer or from the Feature Palette window, and drop it into the drawing window,

– or –

Drag the name from the top of the FeatureManager tree of an open part or assembly document, and drop it into the drawing window.

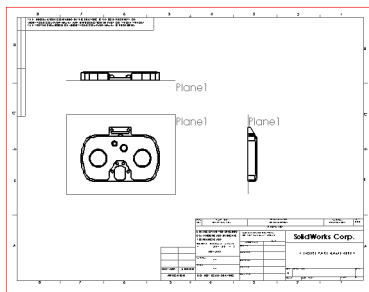
**To create the Standard 3 View from a hyperlink in Internet Explorer:**

- 1 In Internet Explorer (version 4.0 or later), navigate to a location that contains hyperlinks to SolidWorks part files.  
For example, on the SolidWorks web site, in the Support section, navigate to the Solid Modeling Library page. You may also be able to obtain SolidWorks parts from certain third-party vendors.
- 2 Drag the hyperlink from the Internet Explorer window, and drop it in an open drawing window. The **Save As** dialog box appears.
- 3 Navigate to the directory where you want to save the part, enter a new name if desired, and click **Save**.  
The part document is saved locally, and the views of the part are added to the drawing.

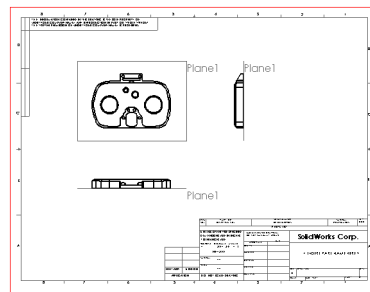
**First Angle and Third Angle Projection**

In third angle projection, the default front view from the part or assembly is displayed at the lower left. In first angle projection, the front view is displayed at the upper left.

See **Modifying the Sheet Setup** on page 9-3 for information about setting the default projection.



3rd Angle Projection



1st Angle Projection



The alignment of the top and side views is fixed in relation to the front view. The top view can be moved vertically, the side view can be moved horizontally.

For more information about arranging views on a sheet, see **Moving Views** on page 9-30 and **Rotating Views** on page 9-31.

## Projection View

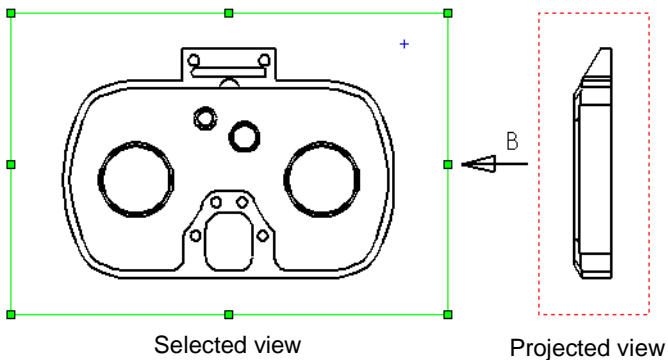
A **Projection** view is created by projecting an orthogonal view, using **First angle** or **Third angle** projection, as specified in the **Sheet Setup** dialog box.

### To create a projection view:

- 1 Select an existing view by clicking inside its boundary.
- 2 Click **Projected View**  on the Drawing toolbar, or click **Insert, Drawing View, Projection**. The pointer shape changes to .
- 3 To select the direction of projection, move the pointer to the appropriate side of the selected view.

A preview of the view is displayed, snapped to the nearest projection. To override the snapping behavior, hold the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

You can project to the left, right, above, or below (in this example, the user moved the pointer to the right of the selected view).




- 4 When the view is where you want it to be, click to place the view.  
The projection view is placed on the sheet, aligned to the view from which it was created. By default, you can move a projection view only in the direction of the projection. See **View Alignment and Display** on page 9-30 for information about changing the alignment of views.
- 5 To display an arrow indicating the direction of projection, right-click the projection view on the sheet or in the FeatureManager design tree, and select **Properties**. Click **Display view arrow** and enter a label if desired (maximum of two characters).

## Auxiliary View



An **Auxiliary** view is similar to a projection view, but unfolded normal to a reference edge in an existing view.

### To create an auxiliary view:

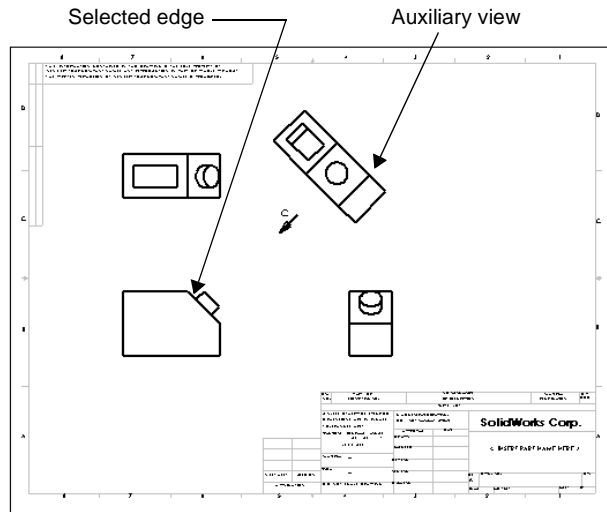
- 1 Select the reference edge (*not* a horizontal or vertical edge, which would create a standard projection view).
- 2 Click **Auxiliary View**  on the Drawing toolbar, or click **Insert, Drawing View, Auxiliary**.

As you move the pointer, a preview of the view is displayed. To override the default alignment as you place the auxiliary view, hold the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

- 3 Move the pointer until the view is where you want it to be, then click to place the view.

The auxiliary view is placed on the sheet, aligned to the edge from which it was created. By default, you can move an auxiliary view only in the direction of the projection. See **View Alignment and Display** on page 9-30 for information about changing the alignment of views


A view arrow (or set of arrows, if you use ANSI dimensioning standards) that indicates the direction of projection is added automatically.



- 4 To remove the arrow, or to change the label, right-click the auxiliary view on the sheet or in the FeatureManager design tree, and select **Properties**. Click **Display view arrow** and enter a label if desired (maximum of two characters).



## Named View

You create a **Named** view by selecting from the view names in the model document. Named views include:

- Standard orthographic views (**Front**, **Top**, **Isometric**, and so on)
- Current** model view
- Custom views that you created by zooming and rotating the model, enabling **Perspective**  if desired, then saving the view by name. The entire model is displayed, even if the selected view orientation displays a partial, zoomed-in view.


For information about creating a named view in the **Orientation** dialog box, see **Using Named Views** on page 1-29.

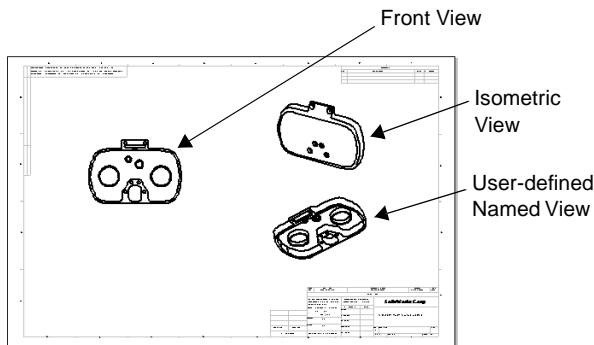
### To insert a named view:

- 1 Click **Named View**  on the Drawing toolbar, or click **Insert, Drawing View, Named View**. The pointer changes to .
- 2 Select the model, using one of the methods described in Step 4 on page 9-12.

You can also right-click in the graphics area of the active drawing window, and select **Insert From File**. Browse to the model document, and click **Open**.

The **Drawing View - Named View** dialog box appears. The list contains the names of all the standard views, the **Current** view, and any custom views you created in the model.

- 3 Double-click a named view from the list, or select a named view and click **OK**. The pointer changes to .
- 4 In the drawing window, click where you want to place the view (a preview of the view is displayed to assist you). Named views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.








## Relative to Model View

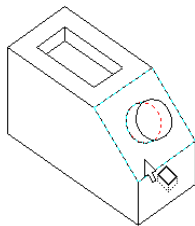


A **Relative to Model** view is an orthographic view defined by two orthogonal faces or planes in the model and the specification of their respective orientations. This view type is also used to set the first orthographic view in a drawing if it is to be something other than a default view.

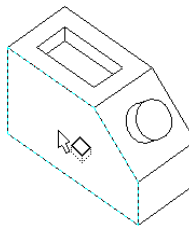
### To insert a relative to model view:

- 1 Click **Relative View**  on the Drawing toolbar, or click **Insert, Drawing View, Relative To Model**. The pointer changes to .
- 2 Select the model, using one of the methods described in Step 4 on page 9-12.  
You can also right-click in the graphics area of the active drawing window, and select **Insert From File**. Browse to the model document, and click **Open**.
- 3 Click a face in the model window that you want to have a specific orientation. The **Drawing View Orientation** dialog box appears.
- 4 Select an orientation (**Front, Top, Left**, etc.), and click **OK**.
- 5 Select another face, *perpendicular* to the first, select the orientation for this face, and click **OK**. The pointer changes to .
- 6 In the drawing window, click where you want to place the view (a preview of the view is displayed to assist you). Relative to model views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

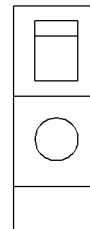
A true view of an angled face perpendicular to the front face of a model is created in this manner. The angled face is specified as **Front** and the front face as **Left**. If the angle of the face in the model changes, the views update to maintain the orientation as originally specified.



Click the angled face, and select **Front**



Click a perpendicular face, and select **Left**



Resulting view, **Relative to Model**

## Detail View

You create a **Detail** view to show a portion of a view, usually at an enlarged scale. This detail may be of an orthographic view, or a 3D view, or a section view.


### To create a detail view:

- 1 Activate an existing view (see page 9-10).

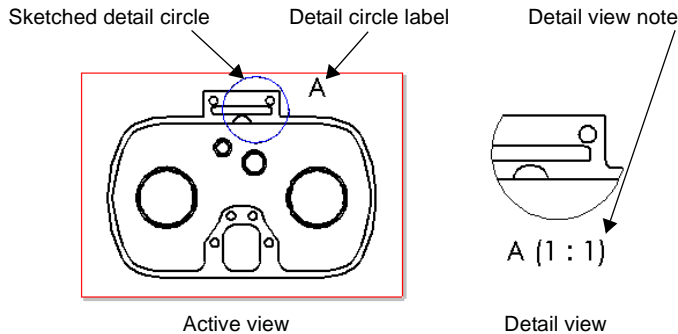
---

**NOTE:** You cannot create a detail view from the following:

- a named view that shows the model with perspective turned on
  - a view that shows an assembly in the exploded state
  - another detail view
- 

- 2 Using a sketch entity tool, create a closed profile around the area to be detailed. (The profile is usually, but not necessarily a circle.) You can add dimensions or relations to the sketch entities to position the profile precisely relative the model.
- 3 Select one entity of the profile.
- 4 Click **Detail View**  on the Drawing toolbar, or click **Insert, Drawing View, Detail**. As you move the pointer, a preview of the view is displayed.
- 5 When the view is where you want it to be, click to place the view.

The note on the detail view includes the letter label and scale.



Detail views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

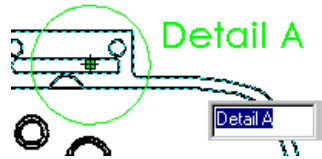
---

**NOTE:** Unexpected lines may appear in a detail view, if sketches are shown in the model. To remove the unwanted lines, edit the model, and hide any sketches that interfere with the detail view.



---

**To modify a detail view:**

- ❑ To change the scale of the detail view, right-click the view on the sheet or in the FeatureManager design tree, and select **Properties**. Under **Scale**, clear the **Use sheet's scale** check box, enter a new scale ratio, and click **OK**.
- ❑ To change the detail view note text, double-click the text (or right-click the text and select **Properties**), and modify it as needed in the **Properties** sheet for the **Note**.
  - If you change the label in the note, the label on the detail profile updates.
  - If you change the scale in the note, the detail view scale updates.
- ❑ To change the text of the label on the detail profile, double-click it, enter the new text, and press **Enter**. The detail view note updates.
- ❑ To change the font and letter of the label, right-click the letter and select **Properties**. Enter a new letter if needed. Clear the **Use Document's Font** check box, click **Font**, then choose a new size or style.
- ❑ To move the note or label on the detail profile, drag the item with the mouse.



**To redefine the cutting edge of the detail profile:**

- 1 Right-click the sketch for the detail profile and select **Edit Sketch**.
- 2 Edit the profile as needed to change the shape or size. The pointer changes to  when it is on the detail profile.
- 3 Select one entity of the profile, and click **Rebuild**  on the Standard toolbar, or click **Edit, Rebuild** to update the detail view.

You can also reshape the detail profile by dragging it. The view automatically updates.


## Broken View

You can use a broken view (or interrupted view) on the drawing of a long part that has a uniform cross-section. This makes it possible to display the part in a larger scale on a smaller size drawing sheet. Reference dimensions and model dimensions associated with the broken area reflect the actual model values.

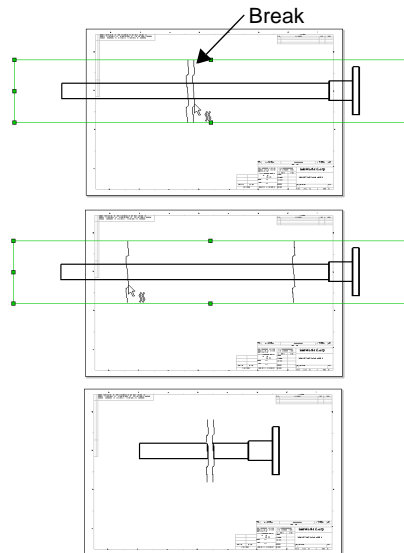
### To create a broken view:

- 1 Select the drawing view and click **Insert, Vertical Break** (or **Horizontal Break**). Two break lines appear in the view.

You can add more than one set of break lines to a view; however, all breaks must be in the same direction. You can only add breaks to an *unbroken* view.

- 2 Drag the break lines with the  pointer to the places on the part where you want the breaks to occur.
- 3 Right-click inside the highlighted boundary and select **Break View**.

The part is displayed with a gap in the part geometry. In addition to model geometry, broken views also support cosmetic threads and axes.




---

**NOTE:** When you break the view, any sketch entities that are between the break lines are deleted. Dimensions that are between the break lines become dangling.

---

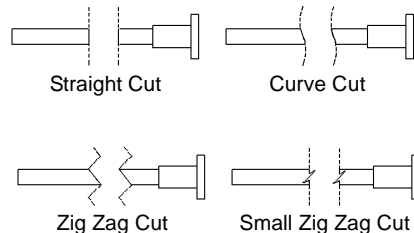
In the drawing view, the break lines extend to the view boundaries to make it easy to select or drag the lines. When you **Print** (or **Print Preview**) the drawing, the break lines are only shown where they actually cut the geometry.

### To restore a broken view to its unbroken state:

Right-click the broken view, and select **Un-Break View**.

### To modify a broken view:

- To change the shape of the break lines, right-click a break line and select a style from the shortcut menu.
- To change the position of the break, drag the break lines.
- To change the width of the break gap, click **Tools, Options, Detailing**. Under **Break Lines**, enter a new value for the **Break Gap**.






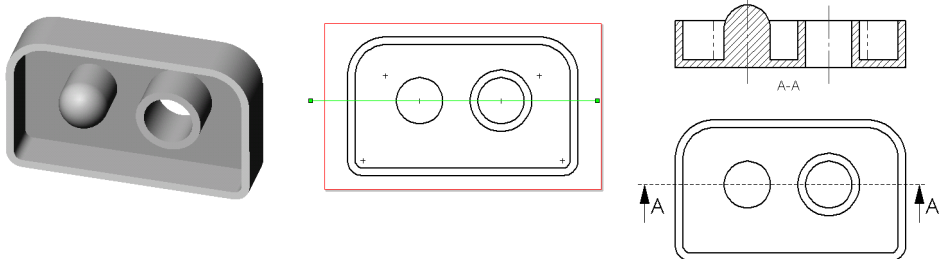
## Section View




You create a **Section** view by partitioning the view with a section line. The view may be a straight cut section or an offset section defined by a stepped section line. The section line may also include arcs.

### To create a section view:

- 1 Activate an existing drawing view (double-click the view, or right-click and select **Activate View**).
  - 2 Click **Centerline**  or **Line**  on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Centerline** or **Line**.
  - 3 Sketch a single or stepped centerline or line through the model in the active view.
  - 4 Select the sketched centerline or line (if it is not already selected). For a stepped line, you only need to select one segment.
  - 5 Click **Insert, Make Section Line**. Notice the arrows, indicating the direction of the cut. If necessary, double-click the section line to reverse the direction of the arrows.
- Now you can edit other section line properties that control the resulting view, if you wish. See **Modifying the section line properties** on page 9-24 for more information.
- 6 With the section line still selected, click **Section View**  on the Drawing toolbar, or click **Insert, Drawing View, Section**.



---

**NOTE:** If you prefer to create the section line and the default section view in a single step, skip Steps 5 and 6. Instead, after you select the sketched segment, click  or **Insert, Drawing View, Section**.

---

As you move the pointer, a preview of the view is displayed. By default, the view is aligned in the direction of the arrows on the section line. To override the default alignment as you place the view, press the **Ctrl** key. To return to the default alignment while dragging, release the **Ctrl** key.

If the section line has multiple segments, the view is aligned to the sketch segment that was selected when you created the section line.



- 7 When the view is where you want it to be, click to place the view.

In its default alignment, a section view can only move in the direction of arrows. See **View Alignment and Display** on page 9-30 for information about changing the alignment of views.

## Aligned Section View

You can create an aligned section view through a model, or portion of a model, that is aligned with a selected section line segment.


### To create an aligned section view:

- 1 Activate an existing drawing view (double-click the view, or right-click and select **Activate View**).
- 2 Click **Centerline**  or **Line**  on the Sketch Tools toolbar, or click **Tools, Sketch Entity, Centerline** or **Line**.
- 3 Sketch the section line. The section line should consist of at least two connected lines at an angle to each other.
- 4 Select the segment of the sketched section line to which you want to align the view.
- 5 Click **Insert, Make Section Line**. Notice the arrows, indicating the direction of the cut. If necessary, double-click the section line to reverse the direction of the arrows.

Now you can edit other section line properties that control the resulting view, if you wish. See **Modifying the section line properties** on page 9-24 for more information.

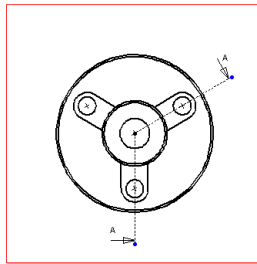
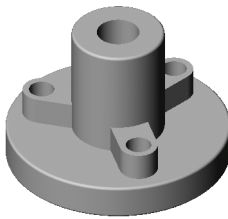
- 6 With the section line still selected, click **Aligned Section View**  on the Drawing toolbar, or click **Insert, Drawing View, Aligned Section**.

---

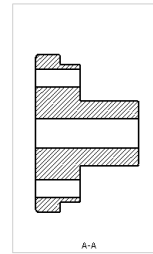
**NOTE:** If you prefer to create the section line and the default aligned section view in a single step, skip Steps 5 and 6. Instead, after you select the sketched section line segment, click  or **Insert, Drawing View, Aligned Section**.

---

You cannot change the angle of the line that you selected to align the view; you can change its position, however.



Active view with section line



Aligned section




As you move the pointer, a preview of the view is displayed. By default, the view is aligned in the direction of the arrows on the selected segment of the section line. To override the default alignment as you place the view, press the **Ctrl** key. To return to the default alignment while dragging, release the **Ctrl** key.

The view is aligned to the sketch segment that was selected when you created the section line; the cut faces from the other segments are projected into the same plane.

- 7 When the view is where you want it to be, click to place the view.

In its default alignment, an aligned section view can only move in the direction of arrows on the selected segment. See **View Alignment and Display** on page 9-30 for information about changing the alignment of views.

## Modifying Section Views

- ❑ To edit the letter label on the section line, double-click the letter label, enter a new letter, and press **Enter**.
- ❑ To edit the note on the section view, double-click the text, make any necessary changes (edit the text, add symbols, change the font, etc.) in the **Note Properties** dialog box, and click **OK**.
- ❑ To edit the location of section lines, drag them as needed.
- ❑ To edit the sketch used to create the section line, right-click an entity of the sketch select **Edit Sketch**. Edit the sketch as needed. Then select one sketch entity, and click **Rebuild**  on the Standard toolbar, or click **Edit, Rebuild** to update the section view.
- ❑ To change the cut direction as indicated by the arrows, double-click anywhere along the section line.
- ❑ After changing the cut direction or editing the section lines, update the view using one of these methods:
  - Click **Rebuild**  on the Standard toolbar, or click **Edit, Rebuild**.
  - Right-click the section view and select **Update View**.
  - Select the view and click **Update View**  on the Drawing toolbar.

## Modifying the crosshatch

The crosshatch pattern specified in *part* documents is used for section views created for the part (and for instances of the part in assemblies). To specify the crosshatch pattern for a part, click **Tools, Options, Crosshatch**. See **Crosshatch Options** on page A-3 for more information.

You can override the pattern for selected faces or components in the section view.

### **To modify the crosshatch pattern for a section view:**

- 1 Right-click the face, or hold **Ctrl** and select multiple faces, and select **Properties**.
- 2 Specify the **Pattern**, **Scale** and **Angle**, and observe the **Preview**.
- 3 Under **Apply To**, specify how to apply the change. Crosshatch patterns are applied in this order of priority:
  - the selected **Region** (highest priority, overrides all other specifications)
  - all the section faces for the selected **Component** (in a section view of an assembly)
  - all the section faces in the **View** (lowest priority, overrides only the document default)

- 4 To *remove* the crosshatch pattern, select the level in the **Apply To** list, and click **Remove Crosshatch**. (This option is only available for levels that can logically be removed.) The pattern is removed one level at a time. For example:
  - If you remove the pattern from a **Region**, the pattern specified for the **Component** is used. If no **Component** pattern is specified, the pattern specified the **View** is used. If no **View** pattern is specified, the document default is used.
  - If you remove the pattern from a **Component**, the pattern for the **View** is used. However, if you specified a pattern for a **Region** on the component, the pattern is retained for the **Region** (a **Region** has a higher priority than a **Component**).
- 5 Click **Apply** or **OK** to accept the changes, or click **Cancel**.

### Modifying the centerline display

Centerlines are automatically added where needed (holes, circular extruded bosses or cuts, and revolved features) in a section view. To specify the distance to extend the centerline beyond the model edge, click **Tools, Options, Detailing**, then enter a value for **Centerline extension**.

You can modify the length of the centerlines by dragging the endpoints. If you do not want the centerlines, you can either delete them from the section view, or you can turn the display of them on and off by clicking **View, Axes**.



### Modifying the section line properties

You can also make changes to a **Section View** or **Aligned Section View** by editing the section line properties. You can set these properties either before or after creating the section view.

To set these properties *before* creating the view, sketch the section line, select it, then click **Insert, Make Section Line**. This makes the sketched line into a section line without automatically creating the associated view. Then right-click the section line and select **Properties**.

To modify the section view *after* it is created, right-click the section line (not the view) and select **Properties**.

After modifying the section line properties, update the section view in one of these ways:

- Click **Rebuild**  on the Standard toolbar, or click **Edit, Rebuild**. (This updates the affected view, plus any other views that need to be updated.)
- Right-click the section view and select **Update view**, or select the view and click **Update View**  on the Drawing toolbar. (This updates only the selected view, so it may be faster.)

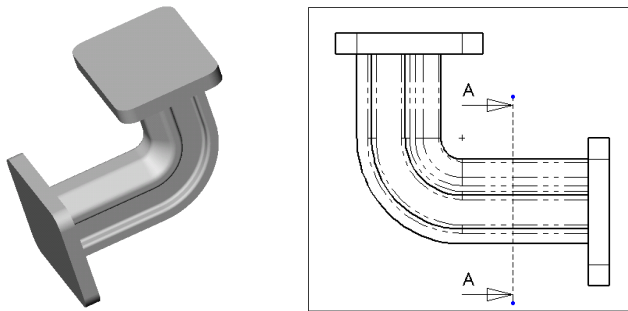


These two options affect only the labels:

- **Label.** Change the letter that is associated with this section line by selecting the letter and entering a new label.
- **Use Document's Font.** When selected, the section line text uses the same font as the rest of the document. When cleared, you can click the **Font** button and specify the font style and size to use.

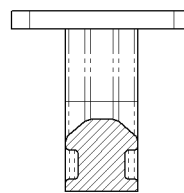
The remaining options in the **Section Line** dialog box control the representation of the model in the resulting view. They may be used individually or in combination, for either section views or aligned section views. Experiment until you get the results that you want. Remember to update the view after you make changes to the **Section Line Properties**.

These examples are based on the model and the view containing the section line shown here.



- **Change direction of cut.** Reverse the direction of the section line cut. This has the same effect as double-clicking the section line.
- **Scale with model changes.** The section line changes size parametrically with the model. This could result in a section cut that is not what you intended, however. This check box is cleared by default.

- **Partial section.** If the section line does not go all the way across the view, selecting this option causes the section view to include only the portion of the model crossed.



Partial section  
*selected*

Change direction of cut  
checked for both views

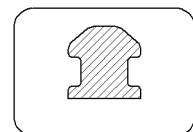


Partial section  
*cleared*

- **Display only surface cut.**  
A simplified view, with surfaces behind the section plane removed. Only the surface(s) actually cut by the section line are displayed in the section view.



Display only surface cut  
*selected*





Display only surface cut  
*cleared*

## Section Views of Assemblies

When you create a section (or aligned section) view of an assembly, you can exclude (leave uncut) selected components by specifying the **Section Scope**.

### To exclude components from a section view of an assembly:

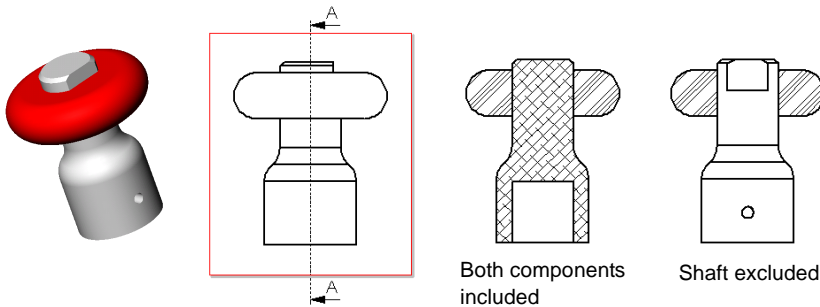
- 1 Activate a view, and create a section line as described in **Section View** on page 9-21 or **Aligned Section View** on page 9-22.
- 2 Click **Section View**  or **Aligned Section View**  on the Drawing toolbar, or click **Insert, Drawing View, Section or Aligned Section**. The **Section Scope** tab of the **Section View** dialog box appears.
- 3 In the active drawing view, or in the FeatureManager design tree, click the components that you want to exclude from the section view. The selected components are listed in the **Excluded components** box.

---

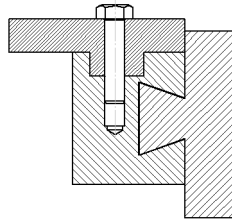
**NOTE:** When selecting components in the FeatureManager design tree, be sure to select them from the *active view*.

---

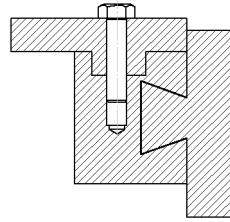
To remove a component from the list, select the component again.



- 4 If a selected component is used more than once in the assembly (for example, if it is a member of a pattern, or if it is used as a component of more than one sub-assembly):
  - To exclude *all the instances* of the selected component, click the component name in the **Excluded components** list, and select the **Don't cut all instances** check box. In the resulting view, all instances of the selected component will be left uncut.
  - To exclude *only the selected instance*, leave this check box cleared. In the resulting view, only the selected instance will be uncut; all others will be cut.
- 5 If you want the software to automatically adjust for neighboring components with the same crosshatch pattern, select the **Auto hatching** check box. The angle used in the pattern automatically alternates in 90 degree increments. If there are still adjacent section faces with the same pattern, the pattern spacing is adjusted.



Auto hatching ON



Auto hatching OFF

If you do not use auto hatching, you can manually adjust the angle and spacing on adjacent components. See **Modifying the crosshatch** on page 9-23.

---

**NOTE:** A detail view created from an assembly section view inherits the crosshatch patterns of its parent view.

---

- 6 Click **OK**. In the resulting section view, excluded components are not cut. They may be shown in the view, or you may remove them from the view by setting the section view property **Display only surface cut**. See **Modifying Section Views** on page 9-23 for more information.

***To edit the section scope for the view:***

Right-click the section view, either on the sheet or in the FeatureManager design tree, select **Properties**, and click the **Section Scope** tab of the **Drawing View Properties** dialog box.

- To add components to the **Excluded components** list, click the components in the section view or the parent view, or in the FeatureManager design tree of the parent view.
- To remove a component from the list, click the component again, or select the name in the **Excluded components** list and press **Delete**.
- To change the display of components that are used more than once in the assembly, select the component in the **Excluded components** list, and select or clear the **Don't cut all instances** check box as needed.

## Troubleshooting Section Views

If a section view is displayed with portions of the model uncut, and with the model in the dangling (brown) color, check for the following conditions:

- The section line may not extend completely across the model. Either extend the section line as needed, or use the **Partial Section** option (see page 9-25).
- The cut may result in zero-thickness geometry that cannot be displayed correctly in the view. Modify the position of the section line to correct this problem.
- There may be a problem with the model geometry. Use **Tools, Check** to identify the invalid geometry. See **Check Entity** on page 5-20 for more information.

## View Properties

---

The **Drawing View Properties** dialog box provides information about the drawing view and the model or assembly that it represents, the view's scale, dimension type, and configuration.

### *To examine or modify drawing view properties:*

Right-click a drawing view on the sheet or in the FeatureManager design tree, and select **Properties**, or select the view and click **Edit, Properties**.

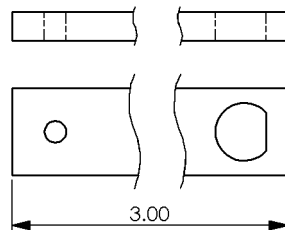
- View information.** Displays the name and type of the selected view (read-only).
- Scale.** To change the scale of the view, clear the **Use sheet's scale** check box, then change the ratio in the **Scale** boxes.
- Model information.** Displays the model name and path to the document (read-only).  
If the selected view is a **Named view**, you can specify a different view orientation. Select a standard or user-defined view from the list.
- Configuration information.** To specify the model configuration to use, click **Use model's "in-use" or last saved configuration**, or click **Use named configuration** then select an existing configuration from the list.

---

**NOTE:** If the model has multiple configurations created with a design table, you can display the design table in the drawing. See **Design Tables in Drawings** on page 8-18.

---

- Show in exploded state.** If the view contains an assembly, and the active configuration of the assembly has an exploded view defined, select this check box to display the exploded view. (See **Exploding an Assembly View** on page 6-35.)
- Type of dimensions created in this view.** To specify the dimension type, select either **Projected** or **True (isometric)**.
- Align breaks with parent.** If the selected view is a broken view, and is the child of another broken view, select this check box to align the break lines in the selected view with the break lines in the parent view. (Click **Edit, Rebuild**, if necessary.)
- Display view arrow.** To display an arrow indicating the direction of projection, select this check box and specify the label (two characters maximum).



## Updating Views

---

You can control the update behavior of the views in a drawing by specifying the **Automatic view update** mode for the active drawing document.

You can also select options that override the current update mode when opening drawings, and you can specify the default update behavior for all new drawings.

### **To change the update mode in the current drawing:**

Right-click the drawing icon at the top of the FeatureManager design tree, and click to either select or clear **Automatic view update**. A check mark beside the menu item indicates that the option is turned on.

### **To specify drawing update options:**

1 Click **Tools, Options, Drawings**.

2 Select from these options:

- **Open existing drawings with automatic view update off.** When *selected*, the software does not attempt to update the views while opening an existing drawing. If the model has changed since the last drawing update, any out-of-date drawing views are marked with a gray crosshatch pattern.

This allows you to open drawings, especially large or complex drawings, more quickly. If **Automatic view update** mode was ON when the drawing was last saved, selecting this option sets the mode to OFF.

When *cleared* (the default), the software updates the views when opening an existing drawing, if **Automatic view update** mode was ON when the drawing was last saved.

- **Set automatic view update for new drawings.** When *selected* (the default), the **Automatic view update** mode is set to ON in all new drawings. When you change the model, the drawing views update automatically.

When *cleared*, the **Automatic view update** mode is set to OFF in all new drawings, and you must update the views manually. You can update all the views at once when you are ready, or update them one by one. When the model changes, any drawing views that need to be updated are marked with a gray crosshatch pattern.

---


**NOTE:** You cannot activate or edit a drawing view that needs to be updated.


---

### **To manually update drawing views:**

1 Turn off **Automatic view update** (click to clear the check mark).

2 To update all the views at once, use one of these methods:

- Click **Edit, Update All Views**.
- Click **Rebuild**  on the Standard toolbar, or click **Edit, Rebuild** to update any views that need to be updated.

- Press **Ctrl-Q** to force a regeneration of the entire drawing.
  - Right-click the drawing icon at the top of the FeatureManager design tree, and click **Update All Views**.
- 3 To update a single drawing view, use one of these methods:
- Select the view, then click **Update View**  on the Drawing toolbar, or click **Edit, Update View**. (Use **Tools, Customize** to add the **Update View** icon to the toolbar, if necessary.)
  - Right-click the view, and select **Update View**.


You can select the view either in the FeatureManager design tree or on the drawing sheet for either method. You also can **Ctrl-click** several views at once for update in this way.

## View Alignment and Display

---

### Moving Views

#### *To move a view:*

Move the pointer over the boundary of a view to highlight the boundary, or select the view. When the move pointer  is displayed, drag the view to a new location. Note the following exceptions:

- Standard 3 View**. The alignment of the front view is fixed in relation to the other two views. When you move it, the other views also move. The other two views can move independently, but only horizontally or vertically, toward or away from the front view.
- Auxiliary, Projection, Section, and Aligned Section** views are aligned with the parent views from which they are created, and only move in the direction of the projection.
- Broken** views respect the alignment of the view before it was broken.
- Named, Detail, Relative to Model, and Empty** views can be moved freely on the sheet; they are not aligned to any other views by default.

#### *To break a view's default alignment:*

For views that have a default alignment, you can break the alignment and move the views independently.

Right-click inside the view boundary (not on the model), and select **Alignment, Break Alignment**, or click **Tools, Align Drawing View, Break Alignment**.

#### *To return a view to its default alignment:*


You can return a view whose alignment you have broken to its original alignment.

Right-click inside the view boundary (not on the model), and select **Alignment, Default Alignment**, or click **Tools, Align Drawing View, Default Alignment**.

## Aligning Views

You can change the alignment of views that are not aligned by default, or views whose alignment you have broken as described on page 9-30.

### *To align one drawing view to another:*

- 1 Select a drawing view, and click **Tools, Align Drawing View, Horizontal to another view** or **Vertical to another view**.  
– or –  
Right-click a drawing view, and select **Alignment, Align Horizontal** or **Align Vertical**.  
The pointer changes to .
- 2 Select the reference view to which you want to align.  
The centers of the views are aligned along the chosen direction. The alignment is maintained if you move the reference view.

To break an alignment created this way, right-click the view and select **Alignment, Break Alignment**, or click **Tools, Align Drawing View, Break Alignment**.

## Rotating Views

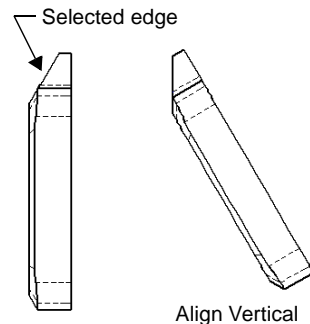
You can rotate a view to orient a selected edge either horizontally or vertically. You can also rotate a view around its centerpoint to orient the view at any angle.

### *To rotate a drawing view:*


- 1 Select a linear model edge in a drawing view.
- 2 Click **Tools, Align Drawing View, Horizontal Edge** or **Vertical Edge**.

The view rotates until the selected edge is oriented horizontally or vertically.

If there are any views projected from a view you change this way, they update to maintain their projection.



### *To rotate a drawing view around its centerpoint:*

Click **Rotate View**  on the View toolbar, then drag the view to the desired rotation. The view snaps to 45° increments, but you can drop the view at any angle.

You can also right-click a view, and select **View, Rotate View** from the shortcut menu. To restore the view to its original rotation, right-click the view and select **Alignment, Default Alignment**. If the default alignment to another view was broken, this also restores the original alignment.




## Copying and Pasting Views

You can cut, copy, and paste drawing views from one sheet to another in the same drawing, and from one drawing document to another.

You can also drag views from sheet to sheet within a drawing (but not to another drawing document).

To perform these operations on multiple views at once, hold down the **Ctrl** key as you select the views.

### **To cut/copy/paste a drawing view:**

- 1 Select the view either on the sheet or in the FeatureManager design tree.
- 2 Click **Cut**  or **Copy** , or click **Edit, Cut** or **Copy**.
- 3 Change to the target sheet, click where you want to paste the view, and click **Paste**  or **Edit, Paste**.

---

**NOTE:** If you want to copy a detail or section view from one drawing to another, you must also copy the parent view from which you created it. You can either copy the parent view first, or you can copy both the parent and the detail/section view at the same time.

---

### **To move or copy views by dragging:**

- 1 Select a view on the active drawing sheet, or select the view's icon in the FeatureManager design tree.
- 2 To *move* the view, hold down the **Shift** key, then drag the selected view or its icon, and drop it on the sheet icon for the target sheet.  
To *copy* the view, hold down the **Ctrl** key as you drag and drop.

## Hiding and Showing Views and Edges

You can use **Hide View** to hide an entire view while working on a drawing. Once hidden, you use **Show View** to display that view again.

When you hide a view that has dependent views (detail, section, or auxiliary views), you are given the choice of hiding those views also. When you show either the parent view or a dependent view again, you are given the choice of showing the related view also.

You also can selectively hide and show edges in a drawing view.




### **To hide/show views:**

- 1 Right-click the view on the sheet or in the FeatureManager design tree, and select **Hide View**.

If the view was active when you hid it, only the active view boundary (gray shadowed) is displayed. Otherwise, the view and boundary are invisible.

To see where hidden views are placed on the sheet without displaying them, click **View, Show Hidden Views**. The boundaries of hidden views are displayed with an **X**.



The pointer changes to  and the view boundary highlights when the pointer passes over a hidden view.

- 2 To show the view again, right-click the view, and select **Show View**.

### **To hide/show edges:**

Right-click an edge in a drawing view, and select **Hide Edge**. The edge is removed from the display, but it still is highlighted when the pointer passes over it.

To display the edge again, right-click the edge, and select **Show Edge**.

If you cannot select the hidden edges, click **Tools, Options, Drawings**, and make sure that the **Select hidden entities** check box is selected.

## **Display Mode**

### **To specify the default display mode for new drawing views:**

- 1 Click **Tools, Options Drawings**.
- 2 Under **Default display for new drawing views**, select **Wireframe**, **Hidden in gray**, or **Hidden lines removed**. This option applies to any *new* views you create; it does not affect any *existing* views.

### **To change the display mode in a view:**

Select a view, then click the appropriate tool on the View toolbar:



**Wireframe**



**Hidden In Gray**

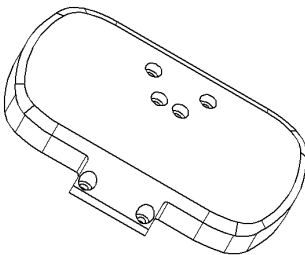


**Hidden Lines Removed**

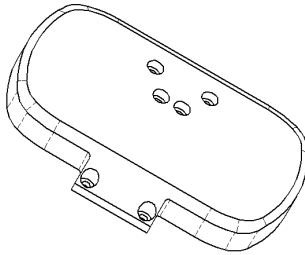
## Tangent Edge Display

**To specify the default display of tangent edges for new drawing views:**

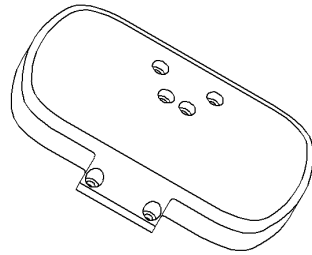
- 1 Click **Tools, Options, Drawings**.
- 2 Select an option for the **Default display for new drawing views**. This option applies to *new* views; it does not affect any *existing* views.
  - **Tangent edges visible** (a solid line)
  - **Tangent edges with font** (as specified on the **Line Font** tab of the **Options** dialog box)
  - **Tangent edges removed**



Tangent edges visible



Tangent edges with font



Tangent edges removed

**To change the display of tangent edges in a drawing view:**

Right-click within the view boundary (not on the model) and select **Tangent Edge**, then **Tangent Edges Visible**, **Tangent Edges with Font**, or **Tangent Edges Removed**.

If you need the ability to select hidden (removed) tangent edges, click **Tools, Options, Drawings**, and make sure that the **Select hidden entities** check box is selected.

## Line Formats and Layers

---

### Component Line Font

You can change the line font for selected components in an assembly drawing.

#### *To change the line font for a selected component:*

- 1 Right-click a component in a drawing view and select **Component Properties**.
- 2 Select **Visible Edges** from the **Type of Edge** list.

If you display hidden lines in drawing views, you can also specify a different font for **Hidden Edges**.




- 3 Select a **Line Style** and **Line Weight**. The line is displayed in the **Preview** box.
- 4 Click **OK**. The new font is used for the edges of the component.

### Line Format

You can specify the color, thickness, and style that is used for any sketch entities that you add to a drawing. You can also modify the appearance of existing lines, edges, and sketch entities in a drawing.

#### *To specify the format for sketch entities in a drawing:*

Click a tool on the Line Format toolbar and change the format as necessary:




- **Line Color.** Click **Line Color** , choose a color from the palette, and click **OK**.
- **Line Thickness.** Click **Line Thickness** , and choose a thickness. As you move the pointer over the menu, the thickness name (**Thin**, **Normal**, **Thick**, etc.) is displayed in the Status bar. Corresponding line weights for printing are defined in **File, Page Setup**, on the **Printer** tab, under **Line weights**.
- **Line Style.** Click **Line Style** , and choose a style. As you move the pointer over the menu, the corresponding line style name (**Solid**, **Dashed**, **Phantom**, etc.) is displayed in the Status bar.

If **By layer** is selected, the properties of the entity are the same as the properties of the active layer.

If **By layer** is cleared, the properties of any entities you create will be different from those specified for the layer.

Sketch entities of any type (lines, centerlines, circles, and so on) that you add to the drawing use the specified font, until you select a different font.

**To change the format of a line, edge, or sketch entity:**

- 1 Select the line, edge, or sketch entity you want to change. To select more than one entity at a time, hold the **Ctrl** key as you select.
- 2 Click a tool on the Line Format toolbar and change the format as necessary. You can change the **Line Color** , **Line Thickness** , or **Line Style** .

To change the line color, line thickness, or line style of the selected entities to match the line format of the layer, select **By layer**.

## Layers


You can create layers in a SolidWorks drawing document. You assign a line color, line thickness, and line style for new entities created on each layer. New entities are automatically added to the active layer. You can hide or show individual layers.

Dimensions and annotations can be moved onto layers; they use the color specified for the layer. Sketch entities use all the properties of the layer.

If you import a **.dxf** or **.dwg** file into a SolidWorks drawing, layers are created automatically. The layer information (names, properties, and entity locations) specified in the system where the **.dxf** or **.dwg** file originated is retained.

If you export a drawing with layers as a **.dxf** or **.dwg** file, the layer information is included in the file. When you open the file in the target system, the entities are on the same layers and have the same properties.

**To create drawing layers:**

- 1 In a drawing, click **Layer properties** .
- 2 Click **New**, and enter the **Name** of a new layer.
- 3 Specify the line format for sketch entities on that layer.
  - To specify the line color, click the **Color** box, select a color, and click **OK**.
  - To specify the line style or thickness, click the **Style** box or the **Width** box, and select the desired font.

An arrow indicates which layer is active. To make a layer become the active layer, double-click beside the layer name.

A yellow light bulb icon appears with any layer that is visible. To hide a layer, double-click its light bulb. The light bulb turns gray, and all the entities on the layer are hidden. To turn the layer back on, double-click the light-bulb again.

To delete a layer, select the layer name and click **Delete**.

To move entities to the active layer, select the entities in the drawing and click **Move**.

## Printing a Drawing

---

You can print or plot an entire drawing sheet, or just a selected area of the sheet. You can print in black and white (the default) or in color.

For basic information about printing and plotting any SolidWorks document, see Chapter 1, “SolidWorks Fundamentals.”

### ***To print a drawing in color:***

- 1 In a drawing, modify the colors of the entities as desired. For information about changing the color, style, and thickness of entities, see **Component Line Font** on page 9-35 and **Layers** on page 9-36.
- 2 Click **File, Page Setup**. On the **Printer** tab, select the **Print drawings in color** check box, and click **OK**.
- 3 Click **File, Print**. In the **Print** dialog box, under **Name**, select a printer that supports color.
- 4 Click **Properties**, verify that any options required for color printing are specified appropriately, then click **OK**. (The options vary among printers.)
- 5 Click **OK** to print.

Any entities that appear in color on the drawing sheet are printed in color. When the specified printer is set up for color printing, **Print Preview** displays the drawing in color also.

### ***To print the whole drawing sheet:***

- 1 Click **File, Print**. In the **Print** dialog box, under **Print range**, click **All** or specify the **Pages** you want to print.
- 2 Under **Scale**, click the **Scale To Fit** check box to print the entire sheet on the page, or clear the check box and specify a **Scale** value (by percent).
- 3 Click **OK**.

### ***To print a selected area of a drawing sheet:***

- 1 Click **File, Print**. In the **Print** dialog box, under **Print range**, click **Selection**, and click **OK**.

The **Print Selection** dialog box appears, and a selection frame is displayed on the drawing sheet. The frame reflects the current printer settings (page size, orientation, etc.) as defined under **File, Page Setup, Print Setup**.

- 2 Select a scale factor to apply to the selected area.
  - **Model scale (1:1)**. This is the default. This means that the selected area is printed at *actual size*; a model dimension of 100mm is printed at 100mm. The default sheet scale is used to calculate the correct print size; therefore, for views that use a scale different than the default sheet scale, you may need to use a custom scale to achieve the desired result.

- **Sheet scale (n:n).** The selected area is printed as it appears on the full sheet. If the drawing size and the paper size are the same, the entire sheet is printed. Otherwise, only the selected area is printed, as it would appear on the full sheet.
- **Custom scale.** The selected area is printed at a scale factor you define. Enter the values you want in the boxes, then click **Apply scale**.

When you change the scale factor, the selection frame size changes accordingly.

- 3 Drag the selection frame to position it over the area you want to print. You can pan or zoom the view, or change the sheet while the selection frame is displayed.
- 4 Click **OK** to print the selected area.

## Sending Drawing Documents

---

You can send drawing documents by electronic mail. You can choose to send only the drawing itself, or you can send the drawing and all referenced documents.

This function uses the electronic mail application installed on your system.

### *To mail a drawing document to another computer:*

- 1 With a drawing document active, click **File, Send To**.
- 2 In the **Send Mail** dialog box, select one of the following:
  - **Drawing\_name.slddrw only**
  - **Drawing\_name.slddrw and other documents it references**

A list of the referenced documents, the number of documents, and the combined size of the documents is displayed.

---

**NOTE:** The size of the combined documents may be important to you because mailing large size documents may cause transmission problems.

---

If you choose to send the drawing and referenced documents, you do not have to send all the documents. You can:

- Click a filename you do not want to send, and click **Remove**.
  - Click a removed filename and click **Attach** to replace the file.
  - Click **Attach All** to return all the removed files.
- 3 Click **OK** to send the documents.
  - 4 Enter your mail password, the mailing address, and any other information requested by your mail application.

## 2D Sketching

---

You can create drawing geometry using 2D sketched geometry only, without reference to existing models or assemblies. This sketched geometry may be controlled by relations (collinear, parallel, tangent, and so on), as well as parametric dimensions.

Before starting to sketch geometry, it is often desirable to display a grid. Right-click the active drawing sheet and select **Display Grid**.

Sketch tools and sketch relations work the same way in a drawing document as they do in a part or assembly document. The only difference is that instead of sketching on model planes or faces, you sketch on the drawing sheet or in an active view.

For information about using sketch tools and sketch relations, see Chapter 2, “Sketching.”

### Empty Drawing View

You can create an empty drawing view to contain your sketch geometry. When this view is activated, all sketch geometry added belongs to this view. The sketch geometry can then be scaled, moved, and deleted as a group while still retaining the editability of the individual sketch entities.

*To create an empty drawing view:*

Click **Insert, Drawing View, Empty**.

### Importing Sketch Geometry into a Drawing

You can import DXF/DWG files into a SolidWorks drawing. Then you can insert that geometry into a sketch to create model features in a part. See **DXF/DWG Files (.dxf, .dwg)** on page 14-13 for more information.

### 2D Emulator

The SolidWorks software includes an add-in application that allows you to create sketch entities by entering commands in text form.

See **2D Sketching and Command Emulator** on page 1-38 and online help for more information.

You can add much of the necessary detailing of your models in the part and assembly documents. This includes dimensions, notes, symbols, etc. Then you can import the dimensions and annotations from the model into a drawing.

Once in the drawing, you can add other annotations, reference dimensions, and a bill of materials if required.

This chapter discusses the following detailing topics:

- Selecting detailing options
- Dimensioning in drawings
- Annotating parts, assemblies, and drawings
- Adding a bill of materials to an assembly drawing



## Detailing Options

---

You have many options for customizing your drawings to meet your company's detailing standards. To select default drawing and detailing options, click **Tools, Options**, then review the options available on these tabs:

- Drawings.** Specify the scale, and various display and update options for views. See **Drawings Options** on page A-8.
- Crosshatch.** Specify the crosshatch patterns used for section views, and for user-defined hatched areas. See **Crosshatch Options** on page A-3.
- Line Font.** Specify the line weights and styles used for various edge types in the drawing. See **Line Font Options** on page A-17.
- Detailing.** Specify numerous options related to dimensions and other types of annotations. See **Detailing Options** on page A-4.

## Annotating Drawing Documents

---

You can add many types of annotations to SolidWorks *part* and *assembly* documents. Additionally, for *drawing* documents, you can:

- Import the annotations, dimensions, and reference geometry from the assembly and part documents into the drawing.**

Changes that you make to imported annotations and dimensions in the part or assembly document are updated in the drawing automatically. Changes you make to imported annotations and driving dimensions in the drawing are updated in the part or assembly document automatically.

---

**NOTE:** One-way associativity can be set between drawings and models during the software installation. This permits changes to the drawing that do not change the model. This option can only be reset with a new installation of the software.

---

The imported items are displayed in the **Imported Annotations** color defined in **Tools, Options, Color** tab. You can drag the attachment point along the edge or face where it is attached, but you cannot re-attach an annotation to a different edge/face/vertex or free space. The attachment of the annotation in the drawing does not affect the attachment in the part or assembly.

For information about importing reference geometry, see **Reference Geometry in a Drawing** on page 10-17.

- Add annotations and reference dimensions in the drawing.**

Annotations and reference dimensions added in a drawing do not affect the part or assembly document. Reference dimensions and annotations are displayed in the **Reference Annotations** color defined in the **Tools, Options, Color** tab.

For more information about creating reference dimensions in a drawing, see **Adding Reference Dimensions** on page 10-5. For more information about creating annotations in any type of SolidWorks document, see **Creating Annotations** on page 10-19.

## Dimensions in a Drawing

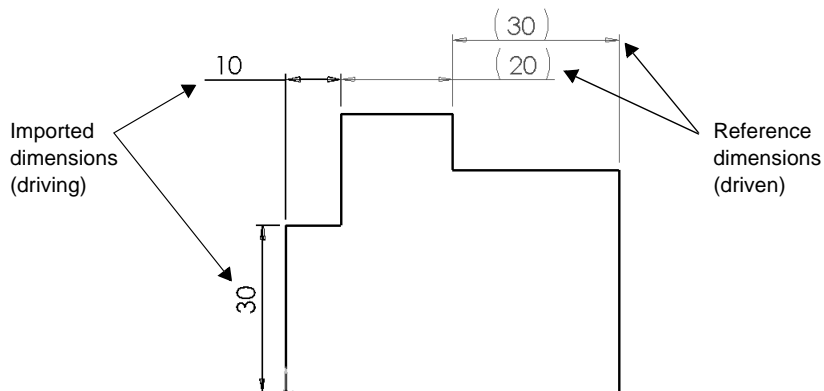
---

Dimensions in a SolidWorks drawing are associated with the model, and changes in the model are reflected in the drawing.

Typically, you create dimensions as you create each part feature, then insert those dimensions into the various drawing views. Changing a dimension in the model updates the drawing, and changing an imported dimension in a drawing changes the model.

You may also add dimensions in the drawing document, but these are *reference* dimensions, and are *driven*; you cannot edit the value of reference dimensions to change the model. However, the values of reference dimensions change when the model dimensions change.

By default, imported dimensions are black. This includes dimensions that are blue in the part or assembly document (such as the extrusion depth). Reference dimensions are gray and appear with parentheses. To display reference dimensions without parentheses, clear the **Add Parentheses by Default** check box on the **Detailing** tab of the **Options** dialog box, or right-click an individual reference dimension and clear the **Show Parentheses** check box.



## Importing Model Dimensions

You can import the dimensions from the model into all the drawing views at once, or into selected drawing views.

- ❑ When you import dimensions *to all views at once*, dimensions appear in the most appropriate view. For example, the diameter of a hole is displayed in its **Normal To** view. Features that appear in partial views, such as **Detail** or **Section** views, are dimensioned in those views first. Inserting dimensions this way reduces the effort required to clean up the drawing.
- ❑ When you import dimensions *to selected views*, you can import the dimensions for the entire model, or you can selectively import the dimensions for one or more components (in an assembly drawing) or features (in a part or assembly drawing).

Dimensions are placed only in the views where they are appropriate. Duplicate dimensions are not imported automatically. Once a dimension has been imported to one view, it is not imported again to another view.

You may delete a dimension from one view, then import it to a different view, or you may move or copy it to another view (see **Moving and Copying Dimensions** on page 10-8).

### ***To insert the model dimensions in a drawing:***

- 1 Select the view(s) to which you want to import dimensions. If you want to import dimensions to all the views at once, do not pre-select a view.
- 2 Click **Insert, Model Items**. The **Insert Model Items** dialog box appears.
- 3 Select **Dimensions**.  
To display the instance count in a pattern or the revolution count in a helix, select **Instance/Revolution Counts**.
- 4 Change the selection of views if necessary. To import dimensions to selected views, clear the **Import Items into All Views** check box, then click the desired views on the drawing sheet or in the FeatureManager design tree.  
The selected views are listed in the **Import into Views** box. Click the views again to remove them from the list; right-click the drawing sheet, and select **Clear selections** to remove all the views from the list.
- 5 If you want to include the dimensions for cosmetic threads that appear as hidden lines, click **Include Dimensions from Hidden Features**.
- 6 Click **OK**.

### ***To insert model dimensions for a selected component or feature in a drawing:***

- 1 In a part drawing, click a face of the feature; in an assembly drawing, click a component or the face of a feature (or expand the **Drawing View** in the FeatureManager design tree, and select the component or feature).
- 2 Click **Insert, Model Items**, and select **Dimensions** (and **Instance/Revolution Counts** if needed).

### 3 Under **Import From**:

- Click **Selected Feature** to add the dimensions for only the selected feature.
- Click **Selected Component** to add the dimensions for the selected component (available when an assembly component, or a feature of an assembly component, is selected).
- Click **Entire Model** to add the dimensions for the entire part or all the assembly components.

---

**NOTE:** When you pre-select a component or feature, dimensions for the item are displayed only in the view where you selected the item. You cannot select a different view or all views.

---

### 4 Click **OK**.

## Adding Reference Dimensions


You can use the same methods to add parallel, horizontal, and vertical reference dimensions to a drawing as you use to dimension sketches. See **Dimensioning Sketches** on page 2-33 for more information. Additionally, you can create *ordinate* or *baseline* dimensions in a drawing.

---

**NOTE:** Reference dimensions are not driving dimensions. You cannot edit reference dimensions to change the model. However, when you change the model, the reference dimensions update accordingly.

---

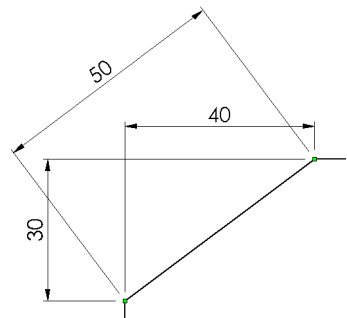
### **To add parallel dimensions to a drawing:**

- 1 Click **Dimension**  on the Sketch Relations toolbar, or click **Tools, Dimensions, Parallel**.
- 2 Click the geometry to dimension. As you move the pointer around the model, a preview of the dimension is displayed.



The position of the pointer relative to the attachment points causes the dimension to snap to the appropriate type (horizontal, vertical, linear, radial, etc.). When the preview indicates the desired dimension type, you can lock the type by right-clicking.

To override the snap behavior *before* selecting the points to dimension, right-click and select either **Horizontal Dimension** or **Vertical Dimension** (or click **Tools, Dimensions, Horizontal** or **Vertical**).


- 3 Click to place the dimension.

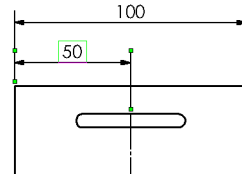


**To dimension a silhouette edge in a drawing:**

- 1 Activate the drawing view where you want to add the dimension.
- 2 Click **Dimension** .
- 3 To create inferences for the edges, point momentarily to a face whose edges appear as silhouettes.
- 4 Point to the silhouette edge, and when the  pointer appears, click to dimension.

**To dimension to a midpoint of a linear edge:**


- 1 Activate the drawing view to dimension.
- 2 Click **Dimension** .
- 3 Right-click the edge whose midpoint you want to dimension, and click **Select Midpoint**.
- 4 Click the second entity to dimension, then click to place the dimension.

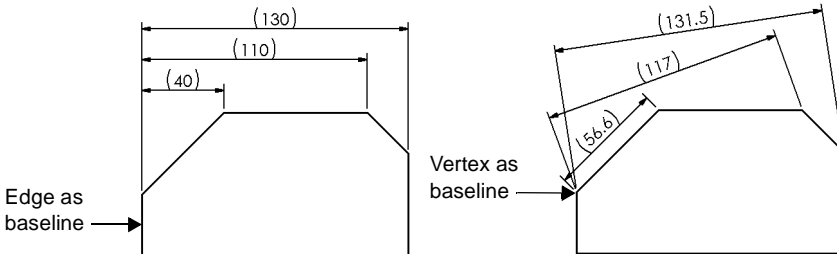


You can also dimension to midpoints when you are adding baseline dimensions or ordinate dimensions (see the sections that follow). Activate the view and select the baseline or zero ordinate. Then follow Step 3 to dimension to midpoints as needed.

## Baseline Dimensions

**To add baseline dimensions to a drawing view:**

- 1 Click **Tools, Dimension, Baseline** (or click **Dimension**  on the Sketch Relations toolbar, then right-click and select **Baseline Dimension**).
- 2 Click the edge or vertex you want to use as a baseline.
  - If you select an *edge*, dimensions are measured *parallel* to the selected edge.
  - If you select a *vertex*, dimensions are measured *point-to-point* from the selected vertex.




- 3 Click the edges or vertices you want to dimension using the same baseline.

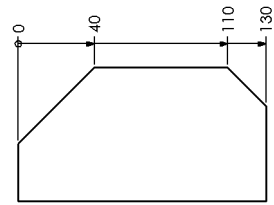
As you click each item, the dimension is placed in the view. The dimensions are aligned using the spacing values specified for **Dimension offset distances** (click **Tools, Options, Detailing, Arrows** to modify the values).

Baseline dimensions are automatically grouped, so that when you drag any member of the group, all the members move together, maintaining the spacing.

## Ordinate Dimensions

### To insert ordinate dimensions:

- 1 Click **Tools, Dimension, Ordinate** or **Horizontal Ordinate** or **Vertical Ordinate** (or click **Dimension**  on the Sketch Relations toolbar, then right-click, and select **Ordinate Dimension**, **Horizontal Ordinate**, or **Vertical Ordinate**).
- 2 Click the entity at the zero position, and click to place the dimension outside the model.
- 3 Click the edges or vertices you want to dimension using the same ordinate. As you click each item, the dimension is placed in the view, aligned to the zero ordinate.



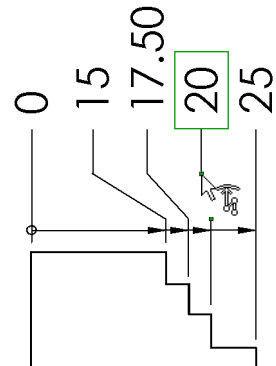
Ordinate dimensions are automatically grouped, so that when you drag any member of the group, all the members move together, maintaining the alignment.

To disconnect a dimension from the alignment group, right-click the dimension, and select **Break Alignment**.

If adjacent dimensions are very close together, the leaders are automatically bent as needed to prevent overlapping text.

Drag handles are displayed at the bends when you select an ordinate dimension with a bent leader.


You can remove the bend, or add a bend to a different ordinate dimension.

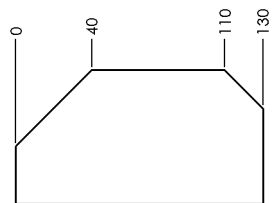


### To jog ordinate dimensions to reduce crowding:

- 1 Right-click the ordinate dimension whose leader you want to bend, and select **Jog**.
- 2 Drag as needed to adjust the space between dimensions. The dimension stays aligned to the others in the ordinate group.
- 3 To return the dimension to its original place, right-click the dimension, and deselect **Jog**.

### To remove (or restore) chain arrows on ordinate dimensions:

- 1 Right-click one of the dimensions in the ordinate group, and select **Properties**.
- 2 Clear (or select) the **Display as chain dimension** check box.
- 3 Click **Rebuild**  to update the display for all the dimensions in the ordinate group.



**To add additional ordinate dimensions:**

- 1 Right-click one of the dimensions in the ordinate group, and select **Add to Ordinate**.
- 2 Select an entity to dimension, and click to place the dimension. The added dimension is aligned and grouped to the others in the same ordinate group.

**To line up ordinate dimensions:**

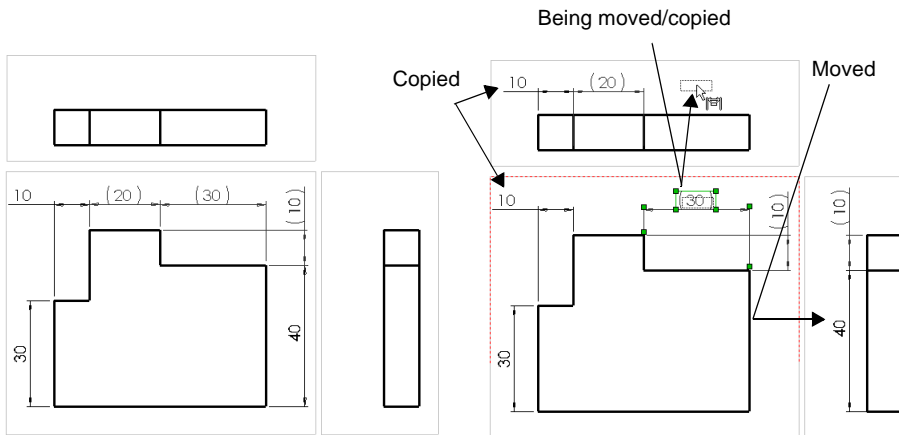
Right-click one of the dimensions in the ordinate group, and select **Align Ordinate**.

Any ordinates that have been disconnected from the group are re-aligned to the zero dimension. However, they are *not* returned to the alignment group. See **Aligning Dimensions** on page 10-9 for more information.

## Moving and Copying Dimensions


Once dimensions are displayed, you can move them within a view or to another view. When you drag a dimension from one place to another, the dimension reattaches to the model, as appropriate. You can only move or copy dimensions to a view where the orientation is appropriate for that dimension.

- To *move* a dimension *within the view*, drag the dimension to the new position.
- To *move* a dimension *from one view to another view*, hold the **Shift** key as you drag the dimension into another view.
- To *copy* a dimension from one view to another view, hold the **Ctrl** key as you drag the dimension into another view.
- To move or copy several dimensions at once, hold the **Ctrl** key as you select.



## Aligning Dimensions

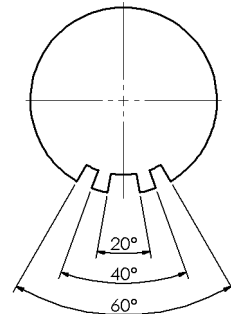
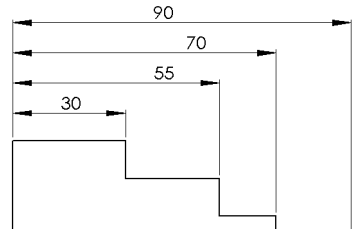
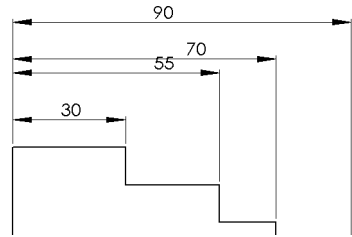
### To distribute parallel dimensions:

- 1 Select a group of dimensions, either by dragging a rectangle around them, or by holding the **Ctrl** key while selecting them. The group can include linear and baseline dimensions, and radial dimensions displayed as linear dimensions.
- 2 Click **Align Parallel/Concentric**  on the Drawing toolbar, or click **Tools, Dimensions, Align Parallel/Concentric**.


The dimensions are distributed with the parallel arrows spaced uniformly. To specify the spacing values, click **Tools, Options**, select the **Detailing** tab, and click the **Arrows** button. Under **Dimension offset distances**, enter values for **From model** (used only for baseline dimensions) and **From last dimension**.

You can also distribute a group of angular dimensions this way, if the angles have a common vertex (concentric arrows are spaced uniformly).

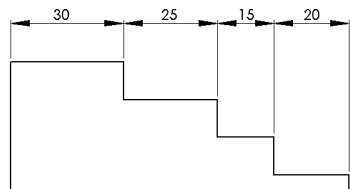
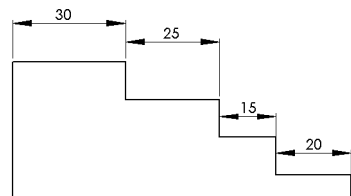
The dimensions are grouped so that they move together when you move any one of them, and the spacing is maintained.



### To align dimensions collinear:

- 1 Select a group of dimensions, either by dragging a rectangle around them, or by holding the **Ctrl** key while selecting them.
- 2 Click **Align Collinear/Radial**  on the Drawing toolbar, or click **Tools, Dimensions, Align Collinear/Radial**.

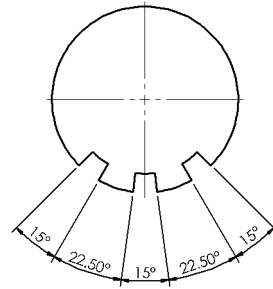
The dimensions are aligned with the arrows collinear, in line with the *last* dimension chosen.





You can also align a group of angular dimensions this way, if the angles have a common centerpoint (concentric arrows are aligned coradial).

The dimensions are grouped so that they move together when you move any one of them.



**To ungroup dimensions that are aligned (baseline, ordinate, parallel or collinear):**

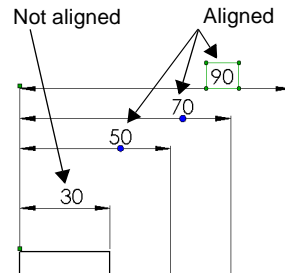
Right-click the dimension to ungroup, and select **Break Alignment**. The selected dimension is disconnected from the group and can move freely; any others that were grouped before remain connected.

To break the alignment of the entire group, select all the dimensions in the group (drag a rectangle around them, or hold the **Ctrl** key while selecting them). Right-click one of them, and select **Break Alignment**.

**To see if a dimension is aligned to others:**

Right-click the dimension you want to know about. If it is a member of an aligned group, you can select **Show Alignment**. A blue dot appears on each of the other dimensions in the aligned group.

If the selected dimension is not a member of an aligned group, the **Show Alignment** item does not appear on the shortcut menu.



**To center dimension text between witness lines:**

Right-click a dimension (linear, radial, or angular), and select **Center Text**.

If the text is inside the witness lines (or inside the circle, in the case of a radial or diameter dimension), it snaps to the center, and is locked in position between the witness lines. If the text is outside the witness lines, it remains outside. To unlock the text position, right-click the dimension, and deselect **Center Text**.

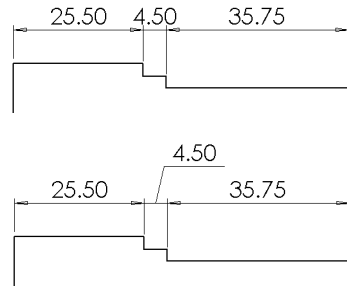
You can also drag the text toward the center of the witness lines to snap it to the center. When you center dimension text this way, its position is not locked between the witness lines, so you can move it freely.

### To offset dimension text from its arrows:

Right-click a dimension (linear, linear radial, or angular), and select **Offset text**.

A leader is attached to the dimension text, and you can drag the text to a new location. When you select an offset dimension, a handle is displayed where the leader attaches to the arrows so you can move the arrows away from the model if needed.

To restore the dimension to its position between the arrows, right-click the dimension again, and deselect **Offset text**.

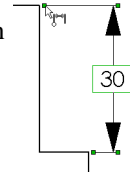


## Modifying Witness Lines

You can change the default attachment point of dimension witness lines, and you can slant the witness lines if needed.

### To change the model attachment point of dimension witness lines:

- 1 Click a dimension. Handles are displayed on the witness and extension lines.
- 2 Drag the handle on the *witness* line (at the attachment point) to the desired position or to a vertex (watch for the inferencing cursor). If you choose a vertex, the default witness line gap is used.

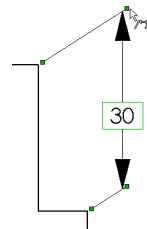


To return the witness line to its original position, right-click the dimension and select **Default Witness Snap**.

### To slant witness lines:

- 1 Click a dimension. Handles are displayed on the witness and extension lines.
- 2 Drag the handle on the *extension* line to slant the witness lines as needed.

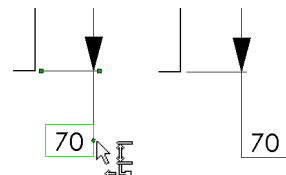
To return the dimension to its original position, right-click the dimension and select **Remove slant**. You can also drag the handle until the dimension snaps back into place in its original position.



### To flip the direction of a leader:

If a vertical dimension is displayed with horizontal text, you can flip the direction of the leader.

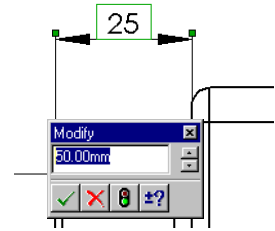
Click the dimension, then click the handle at the corner of the leader.



## Modifying Dimension Values

### *To modify a dimension value in a drawing:*

- 1 Double-click the text of an imported model dimension (not a reference dimension). The dimension value is displayed in the **Modify** box.
- 2 Use the arrows to change the value, or type a new value and press **Enter**.
- 3 Click  to accept the new value and exit without rebuilding.  
Click  to restore the original value and exit.  
Click  to reset the spin box increment.  
Click  to accept the new value and rebuild the model. The **Modify** box remains available, so that you can change the selected dimension again if needed.



---

**NOTE:** If the **Input dimension value** option is checked (under **Tools, Options, General**), the **Modify** dialog box is displayed automatically when you change the attachment of a dimension.

---

## Dimension Properties

You set the default properties for dimensions on the **Detailing** tab of the **Options** dialog box. See **Detailing Options** on page A-4 for more information.

You can modify the properties of a selected dimension at any time during the detailing process.

### *To modify the properties of a dimension:*

- 1 Right-click the dimension, and select **Properties**.  
To select multiple dimensions, hold the **Ctrl** key as you select, then right-click and select **Properties**. You can only edit the properties that can be applied to all the selected dimensions. For example, you cannot edit the **Name**, because names must be unique.
- 2 Change values as described below.
- 3 Click **OK**.

## General Properties

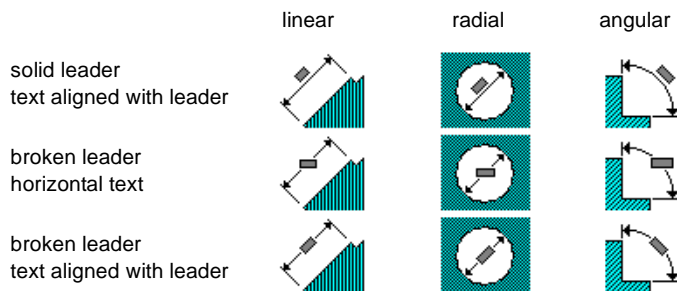
The following properties are common to all types of dimensions.

- **Value.** The numeric value of a dimension.
- **Name.** The name given to a dimension and used in relations associated with the dimension.

- **Full name.** The full name of the dimension, which consists of the **Name**, described above, and the name of the sketch or feature in which it is found (for example, **D2@Sketch1**). The **Full Name** is used in equations and design tables. It is updated when you edit the **Name**.
- **Arrow style.** Either select **Use document's style**, as defined on the **Detailing** tab of the **Options** dialog box, or clear this check box, then choose a different style from the list.
- **Arrows.** Defines placement of arrows with respect to witness lines.
- **Font.** Either select **Use document's font**, as defined on the **Detailing** tab of the **Options** dialog box, or clear this check box, then click **Font**, and select a new style or size.
- **Display precision.** Either select **Use document's precision**, as defined on the **Detailing** tab of the **Options** dialog box, or clear this check box, then click **Precision**, and specify the precision of the primary and alternate units of measure.
- **Display.** Click the button, then select the necessary check boxes in the **Witness/Leader** dialog box to specify which **Witness Lines** and **Leaders** to display.

You can also specify the text alignment with respect to the leaders. To use the default leader and alignment for the dimensioning standard of the drawing, click **Use document's leader display**.

To override the dimensioning standard, select the leader and text alignment you want.



- **Modify Text.** Click the button, then add prefix and suffix text and symbols in the **Modify Text of Dimension** dialog box. See page 10-16.
- **Tolerance.** Click the button, then specify the tolerance display in the **Dimension Tolerance** dialog box. See page 10-16.
- **Units.** To change the units from those specified for the document, clear the **Use document's units** check box, then click the **Units** button. In the **Dimension Units** dialog box, select a different **Length unit** or **Angular unit**, and select other options (**Fractions**, etc.), depending on the type of dimension and units selected.

## Selection-specific Properties

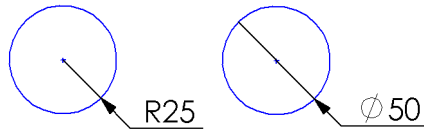
Depending on the type of dimension (linear, radius, reference, driving, etc.) you choose, some of these additional properties may apply.

- **Driven.** Specifies that the dimension is driven by other dimensions and conditions.
- **Read only.** Determines whether feature dimensions can be changed or not.
- **Display as dual dimension.** Displays the dimension in both English and Metric units. The primary unit is specified under **Units** (see the previous section).

If the primary unit is any Metric unit, the secondary unit is decimal inches; if the primary unit is any English unit, the secondary unit is millimeters.

- **Diameter dimension.** Specifies that an arc or circle dimension displays the diameter, not the radius.

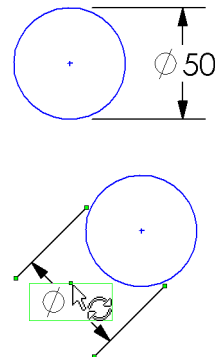
You can also right-click the dimension in a drawing (not in a part or assembly) and select **Display as Radius** or **Display as Diameter** to select or clear this check box.



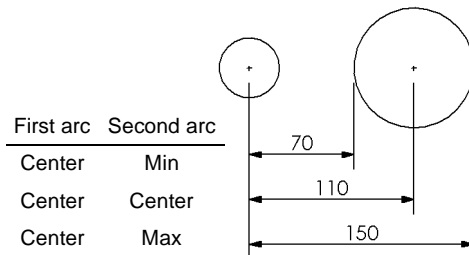
- **Display as linear dimension.** Displays a diameter dimension as a linear dimension, not as a radial dimension.

You can also right-click the dimension in a drawing (not in a part or assembly) and select **Display as Linear** or **Display as Diameter** to select or clear this check box.

If the linear dimension is not placed at the angle you want, select the dimension, and drag the green handle on the dimension text to set the desired position. The dimension snaps in 15° increments around the circle.

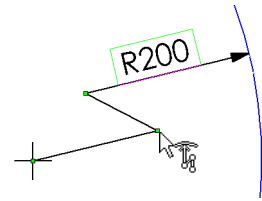


- **First / Second arc condition.** Choose **Center**, **Min**, or **Max** to specify the way to dimension distances between an arc or circle and another entity. See **Dimensioning Arcs and Circles** on page 2-36 for more information.



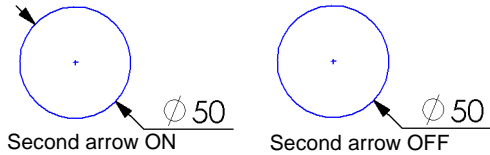
- **Foreshortened radius.** The dimension line for an arc or circle is foreshortened when the centerpoint of the radius is outside of the drawing, or interferes with another drawing view.

To change the angle or length of the segments, click the dimension and drag the handles.



- **Use document's second arrow.**

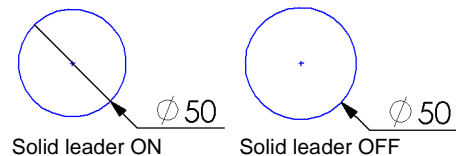
Specifies that the diameter dimension of a circle (displayed as radial, not linear) follows the document default setting for use of a second arrow.



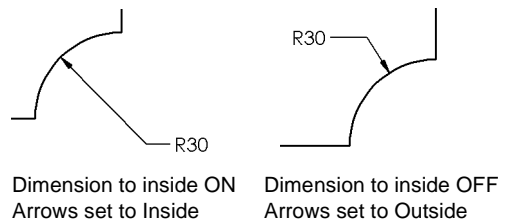
To specify the default, click **Tools, Options, Detailing, Arrows**. To override the default, clear this check box, then use the **Display second outside arrow** check box to turn the arrow display on or off.

- **Display second outside arrow.** A second arrow is used for the diameter dimension with outside arrows. This is only available when the **Use document's second arrow** check box is cleared.

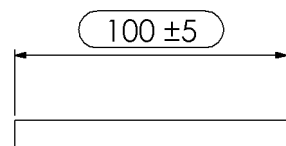
- **Display with solid leader.** Displays a solid line across the diameter dimension of a circle (displayed as radial, not linear) with outside arrows (not available with ANSI standard).



- **Dimension to inside of arc.** Specifies that the dimension arrow is inside the arc. Use this option in combination with the **Arrows** setting (either **Inside** or **Outside**) to meet your drawing standards.



- **Display as chain dimension.** Displays an arrow from the base point to the dimension position on all the dimensions in the ordinate group of the selected dimension. See **Ordinate Dimensions** on page 10-7.
- **Display with parentheses.** Displays the dimension with parentheses, as a reference dimension.
- **Display as inspection dimension.** Displays the dimension inside an oblong. For information about displaying the dimension tolerance, see **Tolerance** on page 10-16.



## Modify Text

You can add a variety of symbols to the text of a dimension.

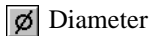
### *To change the text of a dimension:*

- 1 In the **Dimension Properties** dialog box, click **Modify Text**.

The dimension value is represented by **<DIM>** in the middle box. As you make modifications, the text and symbols are displayed in the **Preview** box.

- 2 Click in the **Dimension Text** box where you want to add text or a symbol.
  - To add items to the left or right of the dimension, click in the middle box.
  - To add items above or below the dimension, click in the upper or lower box.
- 3 To add a symbol, click **Add Symbol**. Select a category of symbols from the **Symbol library** list, then select a symbol name, and click **OK**.

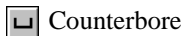
You can add these commonly used symbols by clicking the buttons in the dialog box:



Diameter



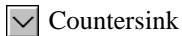
Depth



Counterbore



Square



Countersink

- 4 To replace the dimension value **<DIM>** if it was deleted, click **Add Value**.
- 5 To center the text between the witness lines, click **Center text**. This has the same effect as selecting **Center Text** from the right-mouse menu. See page 10-10.
- 6 Click **OK**.

## Tolerance

### *To modify the tolerance of a dimension:*

- 1 In the **Dimension Properties** dialog box, click **Tolerance**.
- 2 Under **Tolerance Display**, select the type of tolerance to display: **None**, **Basic**, **Bilateral**, **Limit**, **Symmetric**, **MIN**, **MAX** or **Fit**.
- 3 Specify **Maximum / Minimum variation** values appropriate for the type of tolerance that you selected.


**Linear / Angular Tolerance** indicates the type of dimensions to which the current values apply. This is provided for information only.

- 4 Specify the **Font**. Either select **Use dimension's font**, or clear the check box, then specify the **Font Scale** (a number from 0 to 10.0) or **Font Height**.

## Dimension Visibility

You can change the visibility of an individual dimension. Model or reference dimensions may be *hidden* or *shown* to control the appearance of the drawing, temporarily or permanently.

### To hide or show individual dimensions:

- 1 Click **View, Hide/Show Dimensions**. The  pointer appears.
- 2 Click the dimensions that you want to hide or show. Click again to switch between hidden and shown states.

You can also right-click a dimension, and select **Hide** from the shortcut menu.

---

**NOTE:** Any hidden dimensions are displayed in a light gray color when you select **Hide/Show Dimensions**.

---

## Reference Geometry in a Drawing

---

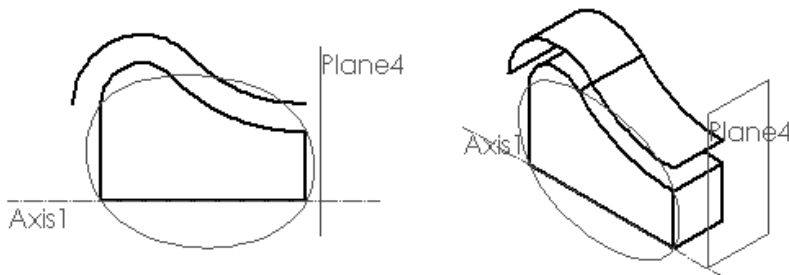
The reference geometry contained in a model is hidden by default in drawing views. When you import the reference geometry, all entities of the selected types are shown. You can import reference geometry to a selected view, or to all views at once, in the same way that you import dimensions. See **Importing Model Dimensions** on page 10-4 for more information.

### To import reference geometry:

- 1 Click **Insert, Model Items**.
- 2 In the **Insert Model Items** dialog box, under **Reference Geometry**, select the types to display (**Axes, Curves, Planes, Surfaces, Points**), then click **OK**.

Planes, axes, curves, and points are shown in gray. Surfaces are shown as visible lines (black).

To toggle the display status of an individual entity, right-click the entity in the FeatureManager design tree, and select **Hide** or **Show**.





## Annotations

---

Annotations behave like dimensions in each type of SolidWorks document. You can add them in the part or assembly document, then import them to the drawing views, or you can create them in the drawing. See **Annotating Drawing Documents** on page 10-2, and **Moving and Copying Dimensions** on page 10-8 for more information about how dimensions and annotations behave.

### Importing Model Annotations

You can import the annotations that belong in a view all at once, or you can selectively import the annotations for a component or feature.

#### *To insert model annotations in a drawing:*

- 1 Select the views you want to annotate. If you want to annotate all the views at once, do not pre-select a view.
- 2 Click **Insert, Model Items**.
- 3 In the **Insert Model Items** dialog box, select the annotation types you want to add, or click **All Types** to import all annotations, dimensions, and reference geometry.
- 4 Click **OK**. The annotations that are appropriate to the selected view are displayed.

#### *To insert model annotations for a selected component or feature in a drawing:*

- 1 In a part drawing, click a face of the feature, or in an assembly drawing, click a component or the face of a feature.  
Alternatively, you can expand the **Drawing View** in the FeatureManager design tree where you want to add the annotations, and select the feature(s).
- 2 Click **Insert, Model Items**, and select the annotation types to add.
- 3 Under **Import From**:
  - Click **Selected Feature** to add the annotations for only the selected feature.
  - Click **Selected Component** to add the annotations for the selected component (an assembly component, or a feature of an assembly component selected).
  - Click **Entire Model** to add the annotations for the entire part or all the assembly components.

---

**NOTE:** When you pre-select a component or feature, annotations for the item are displayed only in the view where you selected the item. You cannot select a different view or all views.

---

- 4 Click **OK**.

## Creating Annotations

You can add *all* types of annotations to a drawing document. You can add *most* types in a part or assembly document, then import them into a drawing document. However, there are some types that you can add *only in a drawing document*. Annotations that are limited to drawing documents are indicated as such in the sections that follow.

To add annotations to any SolidWorks document, click **Insert, Annotations** and select from the available annotation types in the menu.

You can also click one of the icons in the Annotations toolbar. Only those annotation types that are appropriate for the selected entity are available for selection.

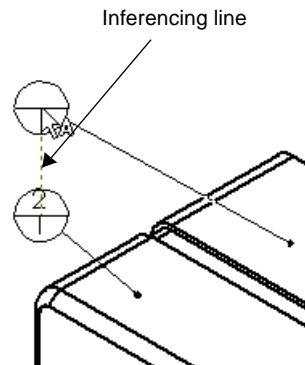


## Aligning Annotations

You can control the snap behavior of detail items that you move. Click **Tools, Options,** and select the **Drawing** tab. Under **Detail Item Snapping,** you have these options:

- **Infer when dragging corner.** When checked, you drag the selected item by the *corner handles* to infer to the *corner* of another detail item.
- **Infer when dragging center.** When checked, you drag the *center* of the item to infer to the *center* of another detail item.
- To ignore all inferences, and to prevent the display of inferencing lines, press the **Alt** key while dragging detail items.

To align an annotation with a neighboring annotation, drag the item (note, balloon callout, and so on) until inferencing lines appear. The item snaps to align with its neighbor.




**Note**



You can create a variety of notes. A note can be free floating or placed with a leader pointing to an item (face, edge, or vertex) in the document. It can contain simple text, symbols, parametric text, and/or hyperlinks.

To specify the default note properties (font, arrow and balloon styles, leader display, and so on), click **Tools, Options, Detailing**.

**To create a note:**

- 1 Select a note location on the drawing sheet, or the attachment point on the model, if the note is to include an attached leader.
- 2 Click **Note**  on the Annotations toolbar, or click **Insert, Annotations, Note**.
- 3 Type the text in the **Note text** box.



To add a *parametric* value to a note, select the corresponding dimension. Parametric text changes when the dimension changes. The dimension *name* is shown in the **Note text** box; the dimension *value* is shown in the note.

For example, you can enter text, then click the dimension for the depth of the base:

**Maximum thickness: "D1 @Base-Extrude@Part1.sldprt" mm.**

The resulting note is:

**Maximum thickness: 20 mm.**

- 4 Modify the **Angle** and **Text justification (Center, Left, Right)** if needed.
- 5 To specify a link to a file or URL, click **Insert Hyperlink** .
- 6 To link the text of the note to either a system-defined property, or to a user-defined custom property, click **Link to Property** . See the next section for more information.
- 7 Select the necessary options for **Leader, Balloon, and Font**.
- 8 To add a modifying symbol, click **Add Symbol**. In the **Symbols** dialog box, select a category of symbols from the **Symbol library** list, then select the symbol, and click **OK**.
- 9 To display the note inside a **Balloon**, select the **Style** and **Size**.
- 10 To specify a font, either select **Use document's font**, or clear the check box, and click **Font**. Select a **Font, Font Style**, and the **Height** in **Points** or in **Units** (the *document* units). If you use **Units**, you can also specify the **Space** between lines (maximum of 1000 units).
- 11 Click **OK**.
- 12 If you selected **Display with leader**, drag the note to the desired location.

You can cut and paste a note and its leader by selecting the note and using **Ctrl-X** and **Ctrl-V**. You can copy and paste using **Ctrl-C** and **Ctrl-V**.

### To edit a note:

- 1 Either double-click the note, or right-click the note and select **Properties** from the shortcut menu.
- 2 Edit the properties as needed, and click **OK**.


## Links to Document Properties

You can link the text of a note to a document property. Properties may be either *system-defined*, or *user-defined* (custom file properties). In either case, if the value of the property changes, the text that is linked to the property changes automatically.

All SolidWorks documents have the system-defined property **SW-File Name** (the name of the document, without the extension). Drawings have these additional system-defined properties: **SW-Sheet Scale**, **SW-Template Size**, **SW-Current Sheet**, **SW-Total Sheets**, and **SW-Sheet Name**.

See **Customizing a Drawing Template** on page 9-5 for information about using these properties to fill in the title block of a drawing automatically.

### To link a note to a property:

- 1 In the **Note** dialog box, click **Link to Property** .
- 2 Select a property from the **Property Name** list. Any properties that have values assigned are included in the list, as well as the system-defined properties for the active document.

To add or change a property, click **Edit Properties**. See **Custom Properties** on page 5-21 for information about how to add properties.

- 3 Click **OK**. The **Note text** shown in the dialog box is the variable name **\$PRP:"propertyname"**. However, in the active document, the note text displays the *value* assigned to the property.
- 4 To link the note to the properties of the model in a drawing view, click **External model reference**, then select a **Property Name** from the list.
  - For a note that belongs to a *view*, the value of the property for the model in that drawing view is displayed. The **Property Name** list includes the properties of the model document, not the drawing document. (If a view is selected when you insert the note, or if the note is attached to the model in a view, the note belongs to the *view*.)
  - For a note that belongs to the *sheet* or the *template*, the value of the property for the model shown in *the first view* inserted into the sheet is displayed by default.

If the sheet contains views of more than one model, you can specify which view contains the model whose properties you want to use. Right-click the sheet, select **Properties**, then select the view from the list under **Use custom property values from model shown in**.


- If no model is shown in the drawing, you can either select a property from the drawing's list, or add a new custom property. The property is resolved when a model is added to the sheet.

5 Click **Rebuild**  to update the drawing, if necessary.

## Hyperlinks

You can add an embedded hyperlink in a note, or you can add a floating hyperlink in any SolidWorks document. The hyperlink can be to a document on the Internet, on your local network, or on your own hard drive.



If the link is to a web site, selecting the hyperlink launches a web browser, if one is not already running, and opens the associated URL (web site). If the link is to a SolidWorks document, clicking the link opens the document. If the link is to another type of document, such as a Word or Excel file, clicking the hyperlink launches the appropriate application, and opens the document.

The actual path of a hyperlink embedded in a note does not appear in the note text. A hyperlinked note is displayed in blue; when the pointer moves over a note that contains a hyperlink, the pointer changes to a hand , and the tooltip indicates the path of the link.

A floating hyperlink (one that is not attached to note text) displays the full path of the link (for example, [HTTP://www.solidworks.com](http://www.solidworks.com)) in blue.


---

**TIP:** Because clicking a hyperlinked note activates the link, moving or editing a hyperlinked note can be difficult.

Move the hyperlink pointer  over the note until a box appears around the text. Point to a corner of the box. When the note pointer  is displayed, drag the note to move it, or right-click the note to edit its properties.

---

### *To create an embedded hyperlink:*

- 1 Create a **Note** as described above, and click **Insert Hyperlink**  in the **Note** dialog box. You can also add a hyperlink while editing a note's properties.
- 2 Enter the path (or browse to the document) to which you want to link the note text. For a World Wide Web location, enter the website name in the format **www.website.com**. (You do not need to enter **HTTP://**)
- 3 Click **Use relative path for hyperlink** if you want to specify a directory path relative to the current location instead of the full pathname.
- 4 Click **OK** to accept the path.



**To create a floating hyperlink:**

- 1 Click **Insert, Hyperlink**.
- 2 Enter the path (or browse to the document) and click **OK**.

 [HTTP://www.solidworks.com](http://www.solidworks.com)

If you edit the properties of a floating hyperlink, you will see that it is a note, the text of which is **\$URL** (a variable representing the path of the link).


**Notes with multiple leaders**

You can select multiple attachment locations for a note. You can add, move, and remove leaders of existing notes.


**To create a note with multiple leaders:**

- 1 Click the attachment points while holding the **Ctrl** key.
- 2 Click **Insert, Annotations, Note**, and proceed as described above.


**To add more leaders to an existing note:**

- 1 With *nothing* selected, move the pointer over the attachment point of a note's leader until you see the  pointer.
- 2 Press the **Ctrl** key, drag the leader to a new attachment point, and release.

**To delete extra leaders:**

- 1 Select the note. The text and the attachment points are highlighted.
- 2 Click the attachment point on the leader to delete, watching for the  pointer.
- 3 Press the **Delete** key.

**To modify the arrows on multiple leaders:**


- 1 Move the pointer over the attachment point of a leader until you see the  pointer, then right-click the leader, and select **Properties**.
- 2 Clear the **Smart** check box if necessary, then choose a new **Arrow style** from the list.
- 3 If you want all the leaders on this note to use the new arrowhead, click **Apply to all arrowheads**. Otherwise the new arrow is applied to only the selected leader.
- 4 Click **OK**.

## Center Mark

*(Drawings only)*

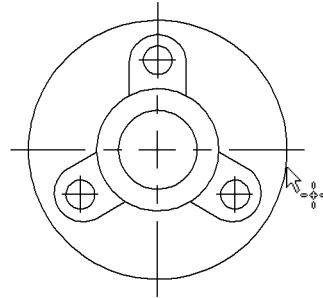
You can place axis lines for showing center marks on circles or arcs that can be used as reference points for dimensioning. You set the default size of center marks and choose whether or not to display extended axis lines on the **Detailing** tab of the **Options** dialog box.

*To create center marks:*

- 1 Click **Center Mark**  on the Annotations toolbar, or **Insert, Annotations, Center Mark**.
- 2 Click the model edges (circles or arcs).

*To change the display attributes of individual center marks:*


- 1 Right-click the center mark and select **Properties**.
- 2 Click to clear **Use document's defaults**.
- 3 Specify the **Mark Size**, select or clear the **Display with lines** check box, and click **OK**.



## Hole Callout

*(Drawings only)*

You can add hole callout symbols to a drawing view.

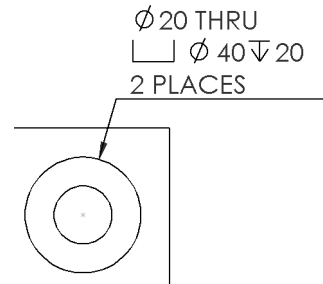
- 1 Click **Hole Callout**  on the Annotations toolbar, or click **Insert, Annotations, Hole Callout**.
- 2 Click the edge of a hole.

The **Modify Text of Dimension** dialog box appears, with symbols appropriate to the selected hole.

You can add text (for example, the number of places where the hole occurs). However, do not change the dimensions or symbols for the size and type of hole in this dialog box.

If you change a hole dimension in the model, the symbol updates automatically.

- 3 Modify as necessary, and click **OK**.



## Cosmetic Thread





You can represent threads on a part, assembly, or drawing, and you can attach a thread callout note.

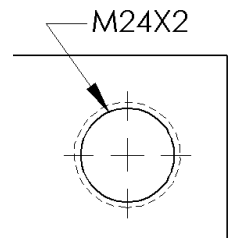
A cosmetic thread differs from other annotations, in that it is an absorbed feature of the item to which it is attached. For example, the cosmetic thread on a hole is in the FeatureManager design tree under the **Hole** feature, along with the sketches used to create the hole.

Cosmetic threads added in a part or assembly can be imported to a drawing view. If you add a cosmetic thread while working in a drawing view, the part or assembly is updated to include a **Cosmetic Thread** feature.

You can specify the line style and weight to use for cosmetic threads on the **Line Font** tab of the **Tools, Options** dialog box.

### **To insert cosmetic threads:**

- 1 On a cylindrical feature (a boss, a cut, or a hole), click the circular edge where the thread begins.
- 2 Click **Cosmetic Thread**  on the Annotations toolbar, or click **Insert, Annotations, Cosmetic Thread**.
- 3 In the **Cosmetic Thread** dialog box, select the thread to apply:
  - Select **Blind** to run the thread for a specified distance, and enter the distance.
  - Select **Up to Next** to run the thread to the next face.
- 4 Enter a value in the **Minor Diameter** (for a boss) or **Major Diameter** (for a cut or hole) box.
- 5 In the **Thread Callout** box, enter thread callout text, if desired.
- 6 Click **OK**, then click **Rebuild** or  on the Standard toolbar to update the part.



### **To edit a cosmetic thread:**

- 1 In a part or assembly document, right-click the **Cosmetic Thread** feature, and select **Edit Definition**. You can select it either in the graphics area, or in the FeatureManager design tree (under the feature to which it is attached).
- 2 Make the necessary changes in the **Cosmetic Thread** dialog box, and click **OK**.

### **To edit the thread callout on a cosmetic thread:**

If you only need to edit the text of the thread callout note, double-click the note (or right-click the note and select **Properties**) in the drawing view, and edit it as needed in the **Properties** sheet for **Note**. See **Note** on page 10-20 for more information.




## Surface Finish Symbol

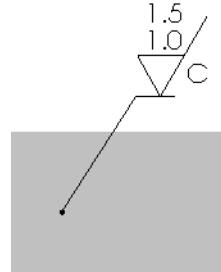


You can specify the surface texture of a part face by using a **Surface Finish Symbol**. You select a model surface, then define a symbol that specifies the values and options for the finish that you want.

### To insert a surface finish symbol:

- 1 Click a face or edge in a part, an assembly, or a drawing view.
- 2 Click **Surface Finish**  on the Annotations toolbar, or click **Insert, Annotations, Surface Finish Symbol**.
- 3 Construct the symbol for the selected surface using the options in the **Surface Finish Symbol** dialog box. As you build the symbol, a preview is displayed.

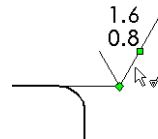
- **Symbol.** Select a symbol type from **Basic**, **Machining Required**, **Machining Prohibited**, or one of the **JIS** types.
- **Direction of lay.** Select the direction of the surface pattern from the list.
- **Roughness.** Specify the **Maximum** and **Minimum** height deviation from the mean plane of the surface, and the **Spacing** between peaks and valleys that form the texture of the surface.
- **Special requirements.** Define any necessary requirements, such as **Production method/treatment**, **Sampling length**, or **Other roughness values**.
- **Material removal.** Specify the **Allowance** (amount of stock to be removed by machining).
- **Leader.** Select **Show** to display the symbol with a leader, and select **Bent leader** if desired. For **Arrow style**, either use the **Smart** option, or clear the **Smart** check box, and choose a different style from the list.
- **Rotated.** Select this check box to rotate the symbol 90°.



- 4 Click **OK** when you are satisfied with the surface finish symbol.

You can drag a surface finish symbol with a leader to any location. If you attach a leaderless symbol to an edge, then drag it off the model edge, an extension line is created.

To edit a surface finish symbol, double-click the symbol (or right-click the symbol and select **Properties**) and make the necessary changes in the **Surface Finish Symbol** dialog box.



Surface finish symbols can have multiple leaders. Adding them is the same as it is for notes. See **Notes with multiple leaders** on page 10-23.



## Datum Feature Symbol



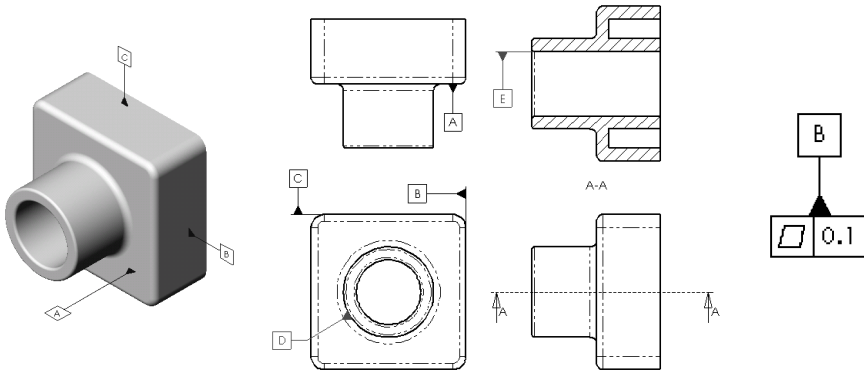
You can attach a datum feature symbol to the following items:

- In a part or assembly, on a planar model surface or on a reference plane
- In a drawing view, on a surface that appears as an edge (not a silhouette) or on a section view surface
- A geometric control frame

### To insert a datum feature symbol:

- 1 Click **Datum Feature Symbol**  on the Annotations toolbar, or **Insert, Annotations, Datum Feature Symbol**. The pointer changes to .
- 2 Click the surface on which you want to place a datum feature symbol.

The datum letters are assigned alphabetically (the letters **I, O, Q, S, X,** and **Z** are omitted in ANSI only).



If you drag a datum feature symbol off the model edge, an extension line is added automatically.

**NOTE:** If you use ANSI dimensioning standards, you can display datum feature symbols in the 1982 style. Click **Tools, Options, Detailing**, and select the **Display datums per 1982** check box.



### To edit the datum feature symbol:


- 1 Double-click the symbol (or right-click the symbol and select **Properties**).
- 2 Edit as needed:
  - Enter a new letter in the **Label** box to change the label.
  - Click **Display with filled triangle** to use a filled triangle at the base; otherwise, an open triangle is used.
  - Click **Display with shoulder** to allow a bent leader.
- 3 Click **OK**.

## Datum Target



You can attach a datum target and symbol to a model face or edge in any document. Both the target (point or area) and the associated symbol are created in one step.

**To create a datum target and symbol:**

- 1 Click a model edge or face.
- 2 Click **Datum Target**  on the Annotations toolbar, or click **Insert, Annotations, Datum Target**.

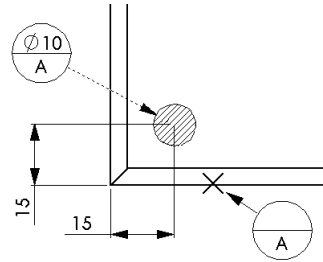
The **Datum Target Symbol** dialog box appears.

- 3 Select a target shape:

- Point**
- Circle**
- Rectangle**

If you choose circle or rectangle, enter the **Target area** size in the boxes.

- 4 Select options as necessary:
  - **Display target area size outside.** Places the size values outside the target symbol balloon.
  - **Display target area.** Displays a crosshatch area of the specified size at the selected location.
  - **Display datum target symbol.** Displays a balloon containing the symbol information.
- 5 Enter the **Datum reference(s)** labels in the boxes.
- 6 Specify the **Leader information**:
  - Select **Bent leader**, if desired.
  - Select **Smart arrow** to use the default attachment arrows, or clear the check box, then choose an **Arrowhead style** from the list.
  - Select a **Line style** (**Solid** or **Dashed**).
- 7 Click **OK**.



To move the datum target symbol (either the balloon or the crosshatch area or point), drag the item.

To edit the datum target symbol, double-click the symbol, or right-click the symbol and select **Properties**. Make any necessary changes as described above.

In a *drawing*, you can dimension to the center of a datum target area. Click either the symbol or the area to add the dimension.

---

**NOTE:** The drawing view must be *active* in order to add dimensions to the symbol.

---

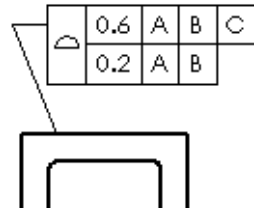
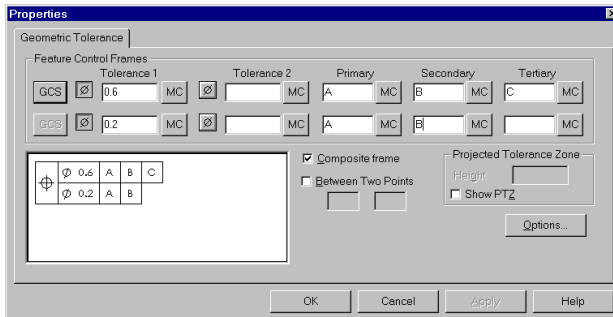
## Geometric Tolerancing Symbol



The SolidWorks software supports the ANSI Y14.5 Geometric and True Position Tolerancing guidelines.

### To create a geometric control frame:

- 1 Select a model entity or a dimension. If you choose an edge, the leader of the frame is placed normal to the selected edge.
- 2 Click **Geometric Tolerance** on the Annotations toolbar, or click **Insert, Annotations, Geometric Tolerance**.
- 3 In the first row of the **Feature control frames** section, click **GCS (Geometric Characteristic Symbol)**. In the **Symbols** dialog box, under **Symbol library**, make sure **Geometric Tolerancing** is selected. Choose a symbol, and click **OK**.  
As you add items, a preview of the resulting symbol is displayed in the box.
- 4 Enter a tolerance value in the **Tolerance 1** box. Click if you want to include a **Diameter** symbol.
- 5 Click **MC**, choose the material condition symbol for **Tolerance 1**, and click **OK**.
- 6 Repeat for **Tolerance 2**.
- 7 Enter tolerance values and material condition symbols for the **Primary**, **Secondary**, and **Tertiary** datums.
- 8 To create a composite frame, enter the values in the second row of the **Feature control frames** section, and click the **Composite frame** check box.
- 9 If the tolerance is between two points, click **Between two points**, and enter the labels of the points in the boxes.
- 10 To display a **Projected tolerance zone**, click the **Show PTZ** check box, and enter a value in the **Height** box.
- 11 To change the **Font** or the style of the **Attached leader**, click the **Options** button. Make the necessary changes in the **Geometric Tolerance Options** dialog box, and click **OK**.
- 12 When the symbol is complete, click **OK**, and drag the symbol to the desired position.




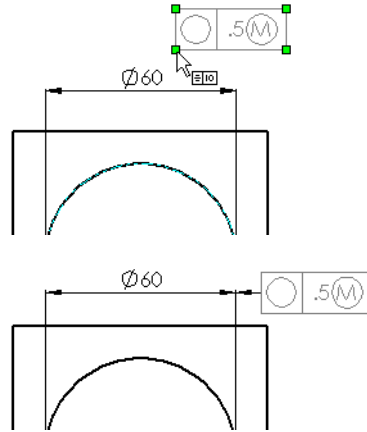
**To edit a tolerance control frame:**

- 1 Double-click the symbol (or right-click the symbol, and select **Properties**).  
The **Geometric Tolerance** dialog box appears with the current values filled in.
- 2 Edit as necessary.
- 3 Click **OK** to update the symbol.

**To attach a geometric tolerance frame to a dimension:**

You can attach a geometric tolerance frame to a dimension in a drawing view.

- 1 Create a geometric tolerance frame as described above. You can attach the symbol to a model entity, or create a floating frame, not attached to anything.
- 2 Select the frame. Note the handles at the corners.
- 3 Drag the frame by a corner handle until the frame is over the dimension value (watch for the dimension cursor ) , and release the mouse button.
- 4 Drag inside the frame to reposition it, if needed. You can place the frame either grouped with the dimension text, or outside the witness lines with a leader.
- 5 To detach the frame, select it and drag a corner handle again. A dimension may have only one attached geometric tolerance frame.



## Bill of Materials (BOM)

### *(Drawings only)*

You can insert a bill of materials into the drawing of an assembly.

---

**NOTE:** You must have the Microsoft® Excel spreadsheet program installed on your computer in order to insert a bill of materials (BOM) into a drawing. It is strongly recommended that you have Excel 97 with Service Release 2 (SR-2).

---

If you select **Automatic update of BOM** in **Tools, Options**, on the **Drawings** tab, assembly changes that affect the BOM are updated automatically. Such changes include adding, deleting, or replacing components, changing component names or custom properties, and so on.

If you do not select **Automatic update of BOM**, you must delete and re-insert the BOM to update it.

#### **To insert a bill of materials:**

- 1 Select a drawing view of an assembly, and click **Insert, Bill of Materials**.

The **Select BOM Template** dialog box appears.

- 2 Select a template and click **Open**. You can use the default template supplied with the software (*install\_directory\lang\english\bomtemp.xls*), or a custom template that you have defined. Custom templates can include special text formatting, additional columns, and so on. See **Custom properties in a bill of materials** on page 10-36 for information about adding custom columns to a BOM.

- 3 In the **Bill of Materials Properties** dialog box, choose from these options:

- **Use Summary Info title as part number.** If you assigned an identifier number in the **Title** box of the **Summary Info** for the component document (under **File, Properties**), use that identifier in the **Part No.** column of the bill of materials. Otherwise, the document name is used.

If the components have multiple configurations, some special rules apply. See **To specify the name to use in the Part No. column:** on page 10-35.

- **Use the document's note font when creating the table.** When selected (the default), the BOM uses the text font specified for notes, under **Tools, Options, Detailing, Note Font**.

If you want to use the fonts specified in the bill of materials template, be sure to clear this option.

4 Choose the way to list sub-assemblies and their components in the bill of materials (for assigning item numbers):

- **Show parts only.** Sub-assemblies are not listed as items; sub-assembly components are listed as individual items.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	2	divider1
4	1	divider2

- **Show top level sub-assemblies and parts only.** Sub-assemblies are listed as items; sub-assembly components are not shown.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	1	divider_assembly

- **Show assemblies and parts in an indented list.** Sub-assemblies are listed as items; sub-assembly components are listed below the sub-assembly item to which they belong, but are not given an item number.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	1	divider_assembly
	2	divider1
	1	divider2

5 Specify the use of an **Anchor Point**:

- **Use table anchor point.** The standard templates supplied with the software include an anchor point for placing the BOM in a specific location on the sheet. To use the point, select this check box, then select which corner of the BOM table to make coincident with the point (top right, bottom left, and so on).

If you do not use the anchor point, the BOM is placed on the sheet near the selected view.

For information about setting the location of the anchor point in the template, see **Customizing a Drawing Template** on page 9-5.

- **Add new item by extending top border of table.** When selected, the BOM shifts upward when new items are added, leaving the bottom row of the table in place. Otherwise, the BOM expands downward when new items are added. This option applies only when you do not use an anchor point.

6 Click **OK**.

**To move the bill of materials:**

If the BOM is attached to an anchor point, you can change which corner of the table is coincident with the anchor point. Right-click the BOM, select **Anchor**, then select a different corner from the list.

You can also detach the BOM from the anchor point, and move it anywhere on the sheet. Right-click the BOM, and select **Anchor, Unlock from Anchor**. Then drag the BOM to a new location.

If you edit the template and move the anchor point, the BOM that is locked to the anchor point moves accordingly.

To attach an unlocked BOM to the anchor point, right-click the BOM, and select **Anchor, Lock to Anchor**.

### **To hide or show the bill of materials:**

In the FeatureManager design tree, expand the drawing view that you selected to create the BOM. Right-click the **Bill of Materials** icon, and select **Hide Bill of Materials**. You can also right-click the BOM table, and select **Hide Bill of Materials**.

To show the BOM again, right-click the BOM icon in the FeatureManager design tree, and select **Show Bill of Materials**.

### **To edit the properties of the bill of materials:**

- 1 Right-click the bill of materials, and select **Properties**. The **Bill of Materials Properties** dialog box appears.
- 2 On the **Configuration** tab, you can modify the options that you specified when you created the BOM (displaying sub-assemblies, using an anchor point, and so on).
- 3 On the **Contents** tab:
  - You can select which items to include in the table. A green check mark in the **ITEM NO.** column means that the item is displayed in the table. Click the check mark to hide the item. This has the same effect as hiding the row in the Excel worksheet.
  - You can move the header row to the bottom of the table, and list the items in order from bottom to top. Clear the **Display labels at top** check box.
- 4 On the **Control** tab:
  - You can specify how **Row numbers** are assigned. If you want to use the item numbers assigned by the software, select the **Row numbers follow assembly ordering** check box.

If you want to assign the item numbers in Excel, and override the system-defined item numbers, clear the **Row numbers follow assembly ordering** check box.

Then specify what happens if a component is removed from the assembly. Under **Reserve row numbers for missing components**, if you select **No**, the rows for the removed components are *deleted* from the table.

If you select **Yes**, specify how the row is displayed: **Display with zero quantity**, **Hide**, or **Display with strike through text**.
  - Under **Split Tables**, you can specify the **Table height before splitting** and the **Split direction for new sections**. If the BOM exceeds the height you specify, the table splits, and continues in another column to the **Left** or **Right**.
- 5 Click **OK**.

### **To edit the text or the formatting in the table:**

- 1 Double-click the BOM. You can also right-click the BOM, and select **Edit Bill of Materials**.

While the bill of materials is active, it is displayed with shaded borders, and row and column headers. Excel toolbars replace the SolidWorks toolbars.

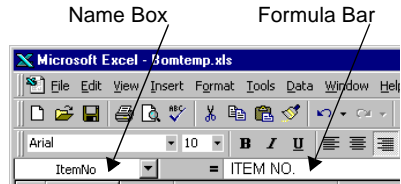


2 Edit the worksheet as needed, using Excel functions. You can perform these kinds of edits:

- Edit the cells in the **Item No.** column or the **Description** column. Before you edit the **Item No.** column, you must clear the **Row numbers follow assembly ordering** check box on the **Control** tab of the **Bill of Materials Properties** dialog box. See the previous section for more information.

If you edit text in the **Qty.** or **Part No.** columns, these changes will be lost the next time that the BOM updates for any reason.

- Rename the column headers. Type the new text in the header cell or in the **Formula Bar**. **DO NOT** edit the text in the **Name box**.
- Add columns or rows, before the cell marked **\$\$END** (but not above the header row). You can include rows for bulk items, such as oil or adhesive.



- Change the order of the columns. Insert a new column, cut and paste the existing columns until they are arranged as needed, then delete the empty column.
- Hide columns. Right-click the lettered cell at the top of the column, and select **Hide**. If the BOM includes a column that you do not want to display, do not delete it; hide it instead.
- Hide rows. Right-click the numbered cell at the left of the row, and select **Hide**. This has the same effect as clearing the check mark beside the item on the **Contents** tab of the **Bill of Materials Properties** dialog box.

Do not delete rows to remove them from the table. If you delete a row in Excel, the row is replaced the next time the BOM is updated.

You cannot delete or hide the header row.

- Change the text alignment (left, right, center) by column.
- Adjust the column width or row height.
- Change the text font.
- Some editing operations apply to an entire row or column in the table. To select an entire row, click the numbered cell at the left of the row. To select an entire column, click the lettered cell above the column.

---


**NOTE:** The following types of cell formatting are not supported in a bill of materials: cell shading (colors and patterns), borders, text orientation (text at an angle), wrap text, merge cells.

---

3 Click outside the worksheet to return to editing the drawing.

You can display a name other than the document name in the part number column of the BOM. For example, you can use a stock number specified by your company, or the configuration name for a part or sub-assembly that has more than one configuration. To do so, it may be necessary to edit the **Advanced** properties of the configuration. See the next section for more information.

**To specify the name to use in the Part No. column:**

- 1 In the part or sub-assembly document, click the Configuration tab  at the lower left of the window to change to the ConfigurationManager view.
- 2 Right-click the configuration you want to edit, and select **Properties**.
- 3 Click **Advanced**. Under **Part No. displayed when used in a Bill of Materials**, select one of the following:
  - **Use document name**. The name of the component document is used.
  - **Use name below**. This option lets you specify a different name to use. The configuration name is displayed by default. Alternatively, you can enter a name of your own choosing.
- 4 If you *always* want a sub-assembly to be listed as a single component in the bill of materials, select the **Don't show child components in BOM when used as sub-assembly** check box.  
  
When cleared, child components are listed individually in the BOM, when **Show parts only** or **Show assemblies and parts in an indented list** is selected (see page 10-31).
- 5 Click **OK**.

**Multiple component configurations in a BOM**

You can specify how to list component configurations in a BOM. When multiple configurations of the same component are used in an assembly, there are some additional considerations.

- In the **Item No.** column, you can list the configurations as a separate entries with individual item numbers, or you can group all the configurations as a single entry with one item number.
- In the **Part No.** column, you can display a user-defined name, the configuration name, or the document name for each item.

In the component document, you need to perform certain steps to achieve the desired results. The steps vary, depending on how you create the configurations.

**If you use a design table to create configurations:**

Item No.	Part No.	Procedure
each configuration as a separate item	user-defined names	Use <b>\$PARTNUMBER</b> <sup>1</sup> in the table, with a different value for each configuration.
	configuration names	This is the default. Design tables automatically set the <b>Advanced Properties</b> <sup>2</sup> to use the configuration name.
all configurations as a single item	a user-defined name	Use <b>\$PARTNUMBER</b> <sup>1</sup> in the table, with the same value for all configurations.
	the document name	Use <b>\$PARTNUMBER</b> <sup>1</sup> in the table, with the document name for all configurations.

***If you create configurations manually:***

Item No.	Part No.	Procedure
each configuration as a separate item	user-defined names	Under <b>Advanced Properties</b> <sup>2</sup> , click <b>Use name below</b> , and specify a different value for each configuration.
	configuration names	Under <b>Advanced Properties</b> <sup>2</sup> , click <b>Use name below</b> . The configuration name is the default.
all configurations as a single item	a user-defined name	<ol style="list-style-type: none"> <li>1 Use the default <b>Advanced Properties</b><sup>2</sup> (<b>Use document name</b> selected).</li> <li>2 Click <b>File, Properties</b>, and enter a <b>Title</b> in the <b>Summary Info</b> dialog box.</li> <li>3 When you create the BOM, select <b>Use Summary Info title as part number</b>.</li> </ol>
	the document name	This is the default for the <b>Advanced Properties</b> <sup>2</sup> ( <b>Use document name</b> selected).

<sup>1</sup> For information about using the \$PARTNUMBER parameter in a design table, see **Part number** on page 8-15.

<sup>2</sup> To edit the **Advanced Properties**, right-click the configuration name, select **Properties**, and click the **Advanced** button.

When the **Advanced Properties** are not defined in the same way for all the configurations of the components, the following order of precedence applies:

- User-defined name or configuration name (if **Use name below** is selected)
- **Summary Info Title** (if the option is selected, and if **Title** is defined for the component)
- Document name

***To save a bill of materials as an Excel file:***

- 1 Click the bill of materials on the drawing sheet.
- 2 Click **File, Save As**. The **Save as type** is set to **Excel Files (\*.xls)** by default.
- 3 Enter a file name, and click **Save**.

You can use the saved bill of materials in Excel, or as an OLE object in other applications. The saved file is *not linked* to the bill of materials in the drawing; edits to one do not affect the other.

**Custom properties in a bill of materials**

You can use *custom properties* to add more information to a part or sub-assembly document. Then you can add columns for these properties to the bill of materials template found in *install\_directory\lang\English\Bomtemp.xls*.

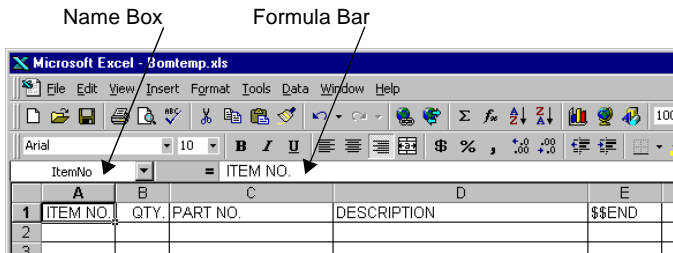
**To add custom columns to the bill of materials:**

- 1 Add custom properties to the part or sub-assembly files, or to the configurations within the file. See **Custom Properties** on page 5-21 for information about how to add custom properties.
- 2 Open the BOM template file. To add the columns to the default BOM template, open **install\_directory\lang\english\Bomtemp.xls**.

To create a new BOM template, it is a good idea to make a copy of the default template, then open the copy.

- 3 Insert a column in the worksheet where you want the custom property to be listed.
- 4 In the top cell in the new column, enter the text you want to display in the column header cell, and press **Enter**. The name appears in the **Formula Bar**.
- 5 Click the cell to select it, then click **Insert, Name, Define**. Enter the name of the custom property (no spaces in the name), and click **OK**. The name appears in the **Name Box**.

The name in the **Name Box** must *exactly* match the name of the custom property in the part or sub-assembly documents. The text in the **Formula Bar** does not have to match the property name.




---

**NOTE:** DO NOT edit the name in the **Name Box** for these columns: ITEM NO. (**ItemNo**), QTY. (**quantity**), PART NO. (**partno**).

You may edit the text that appears in the header cells for these columns (in the **Formula Bar**).

---

- 6 Click **Add**, then click **OK**. A bill of materials that uses this template will now include the added column, with the value specified in the custom properties of the part or sub-assembly document.

This example uses the custom property **Description**, part numbers were entered in the **Title** box of the file's **Summary Info (File, Properties)**, and **Use Summary Info title as part number** was checked.

ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	240-5892	bracket
2	1	N/A	crank assembly
3	1	001-5432	yoke, male
4	1	001-5431	yoke, female
5	1	003-7790	spider
6	1	120-1121	pin, long
7	2	120-1104	pin, short

## Balloon Callout



You can create balloon callouts in a drawing document. The balloons label the parts in the assembly and relate them to item numbers on the bill of materials (BOM).


However, you do not have to insert a BOM in order to add callouts. If the drawing has no BOM, the item numbers are the default values that the software would use if you did insert a BOM. If there is no BOM on the active sheet, but there is a BOM on another sheet, the numbers from that BOM are used.

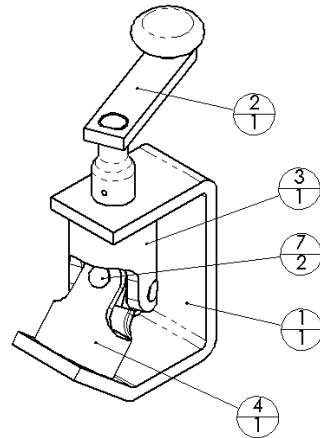
To set the default BOM balloon properties, click **Tools, Options, Detailing, Balloons**, and select the desired options under **BOM Balloons**. If the default BOM balloon **Style** is **Circular Split Line**, you can choose what to display in the **Upper** and **Lower** portions of the balloon: **Item Number**, **Quantity**, or **Custom**.

Balloons are automatically suppressed when the components they reference are suppressed.

Balloons may also be added in assembly documents. To import balloons from an assembly document into a drawing view, select **Notes** in the **Insert Models Items** dialog box.

### **To insert balloon callouts:**

- 1 Click  on the Annotations toolbar, or **Insert, Annotations, Balloon**.
- 2 Click a component in a drawing view of an assembly, or click a component in an assembly model.  
A balloon containing an item number attaches to the part. The number corresponds to the item number in the bill of materials.
- 3 To move the balloon or leader arrow, select and drag the balloon, or drag the leader by the handle.



### **To change the balloon properties:**


- 1 Double-click the balloon, or right-click the balloon and select **Properties**.
- 2 Make the changes that you want in the **Note** dialog and click **OK**.

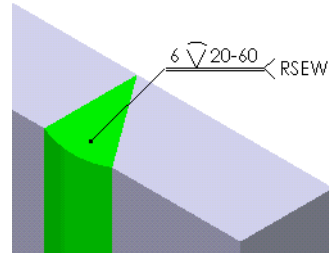
## Weld Symbol

When you create a weld bead in an assembly, a weld symbol is created automatically on the **WeldBead** component, representing the parameters you specified. You can edit this symbol to add more information to it. You can also construct weld symbols independently in any SolidWorks document.

For information about creating weld beads, see **Welding** on page 7-26.

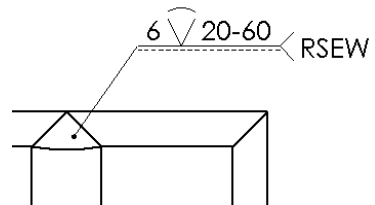
### **To insert a weld symbol:**

- 1 Click a face or edge where you want to indicate a welded joint.
- 2 Click **Insert, Annotations, Weld Symbol** or click  on the Annotations toolbar. The **Properties** sheet for **Weld Symbol** appears.
- 3 In the **Dimension** box, enter a value for the weld dimension.
- 4 Click the **Weld Symbol** button and select a weld type from the list in the **Symbols** dialog box.
- 5 Enter additional dimensions or values as needed for this symbol in the box to the right of the **Weld Symbol** button.



In this example, the length and the pitch of the weld is entered in the box.

- 6 Continue constructing the weld symbol by choosing options from the dialog box. A preview of the symbol is displayed as you add more information.



The options available depend on the dimensioning standard you use (ISO, ANSI, etc.).

You can specify symbols and values for both sides of the reference line. Click **Other Side** (ANSI standard) or **Top** (all other standards), and specify the symbols and values to display above the reference line. Then click **Arrow Side** or **Bottom**, and specify the items to display below the reference line.

### **To edit a weld symbol:**

Right-click the weld symbol, and select **Properties**. Edit as needed, and click **OK**.

Weld symbols can have multiple leaders. Adding them is the same as it is for notes. See **Notes with multiple leaders** on page 10-23.

## Area Hatch

### *(Drawings only)*

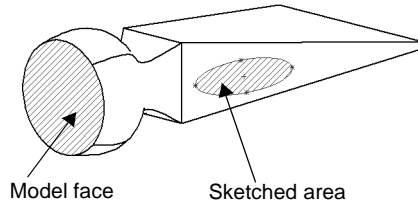
You can apply a crosshatch pattern to a selected face, or to a closed sketch loop.

#### **To add crosshatch to a model face:**

- 1 Activate a drawing view.
- 2 Select a model face.
- 3 Click **Insert, Area Hatch**.

The selected face is filled, using the default crosshatch pattern specified under **Tools, Options, Crosshatch**. See **Crosshatch Options** on page A-3 for more information.

To delete the crosshatch pattern from a model face, select the crosshatch, and press the **Delete** key.



#### **To add a crosshatch area to a sketched area:**

- 1 Sketch a closed profile.
- 2 Select one segment of the profile.
- 3 Click **Insert, Area Hatch**.

You can change the shape and size of the sketched profile as needed. The crosshatch updates to fill the modified shape.

To delete the crosshatch pattern, select the crosshatch, and press the **Delete** key. The sketched profile remains.

#### **To edit the crosshatch pattern:**

- 1 Select the crosshatch area, and click **Edit, Properties**,  
– or –  
Right-click the area and select **Properties**.
- 2 Choose a different **Pattern** and adjust the **Scale** and **Angle** if needed.
- 3 Click **OK**.

## Block

### *(Drawings only)*

You can create, save, and insert custom symbols, called blocks, for drawing items that you use often, such as standard notes, label positions, etc. Blocks can include text, any type of sketch entity except points, and area hatch (see the previous section).


If you store your blocks in a specific folder, you can specify that folder as the default search path for adding blocks.

### **To specify a default path for locating blocks:**

- 1 Click **Tools, Options, External References**.
- 2 Under **Show folders for**, select **Blocks** from the list, then click **Add**.
- 3 In the **Choose Directory** dialog box, locate the desired directory, and click **OK**.

The list can include more than one directory; the *first* path in the list is displayed by default in the **Open** dialog box when you insert a block.

### **To create a block:**

- 1 In a drawing, click **Tools, Block, New**. A block editing window opens.
- 2 Sketch the entities that make up the block. To add text, use the **Note** tool  (see **Note** on page 10-20). You can also insert an existing block in the block you are creating.
- 3 To specify the location for attaching a leader to the block, right-click a blank area, and select **Make Block**. The entities are grouped, and a leader attachment point is placed at the lower left corner of an imaginary rectangle that includes all the entities.

If you want the leader to be attached in a different place, click the block to display the rectangle and a green handle at the default location. Hold the **Ctrl** key, and drag the handle to a new location.

- 4 To save the block, click **File, Save**. In the **Save As** dialog box, navigate to the directory where you want to store the block, and enter a filename (the extension **.sldsym** is added automatically).

### **To insert a block:**

- 1 In a drawing, click the sheet, view, or model where you want to add the block, then click **Insert, Annotations, Block**.
- 2 In the **Open** dialog box, select the block document (**.sldsym**) and click **Open**. The block is copied to the sheet, with the leader attachment point where you clicked.  
If you pre-selected a model, the block includes a leader automatically. If you did not pre-select a model, you can add a leader.
- 3 To modify the way the block is displayed, either double-click the block, or right-click the block and select **Properties**.



- **Attached Leader.** Click **Display with Leader**, and **Display with Bent Leader** if desired, and choose a different **Arrow** style, if desired.
- **Scaling.** Move the slider, or enter a scale value in the box.
- **Rotation.** To rotate the block up to 180 degrees, enter an angle in the box, or move the slider.

To rotate clockwise, enter a *negative* angle value, or move the slider to the left.

To rotate counter-clockwise, enter a *positive* angle value, or move the slider to the right.

- 4 Click **OK**. If you selected an attached leader, drag the block to extend the leader.

### **To edit a block:**

- 1 If a block includes text, you can edit the text after the block is added to the drawing. Double-click the text, type the new text, and press **Enter**.
- 2 To edit the sketch entities of a block, click **Tools, Block, Edit**.
- 3 In the **Open** dialog box, navigate to the directory that contains the block, select the block document (**.sldsym**) and click **Open**.
- 4 Right-click the block in the symbol editing window, and select **Explode Block**. The block items are ungrouped, and you can edit the sketch entities or text as needed.
- 5 When you are finished, click **Save** to save the block with the same name, or **Save as** to save the block with a new name. All the sketch entities in the window are grouped to create the block.

---

**NOTE:** Changes that you make to the block text while working in the drawing are not applied to the original block file. Likewise, changes you make in the block editing window are *not* reflected in any existing instances of the block in drawings. Blocks are *embedded* in drawings; the files are *not linked*.

---

## Design Table

If a model document uses a design table to generate multiple configurations, you can display the table in the drawing. A single drawing can specify the dimensions for all the configurations.

You can label the dimensions with letters or short, descriptive names, instead of using the full dimension names and the values. For example, instead of **D1@BaseExtrude**, you can use a label such as **A** or **thickness** in both the table and the drawing view.

For more information, see **Design Tables in Drawings** on page 8-18.

# Library Features and the Feature Palette Window

---

A *library feature* is a frequently used feature, or combination of features, that you create once and then save in a library for future use.

You can create commonly used features, such as holes or slots, in common sizes, and save them as library features to use whenever you need them. You can use several library features as building blocks to construct a single part. This can save time, and it can help ensure consistency in your models.

The Feature Palette™ window helps you organize and use library features, sheet metal forming tools, piping components, and other commonly used parts. The Feature Palette window gives you quick access to the items you use most, by simply dragging them into the SolidWorks window.

This chapter describes:

- Creating and editing library features
- Adding library features to parts
- Organizing library features, forming tools, piping components, and other parts in the Feature Palette window
- Inserting palette items in parts and assemblies

## Library Features

---

A library feature can contain one or more features. Most feature types are supported, some with certain limitations.

You can edit a library feature once it has been inserted into a part. After a library feature is added to a part, there is no associativity between the target part and the library feature part. If you edit a library feature part, the change is not propagated to parts that contain the library feature.

You cannot insert a library feature that includes a base feature into a part that already has a base feature (you cannot have two base features in a single part). Library features usually consist of features added to a base feature, but not the base feature itself.

You can only add a library feature to a part, or while editing a part in the context of an assembly. You cannot add a library feature to an assembly itself.

### Creating a Library Feature

To create a library feature, you first create a base feature, then create the features that you want included in the library feature on the base. Library features have the **.sldlfp** extension.

#### *To create a library feature:*

- 1 Open a new part, sketch a profile, and create a base feature.
- 2 Create the features you want to have in the library feature.

Dimension the features to the base if you want to use the dimensions to locate the library feature when you place it on the target part.

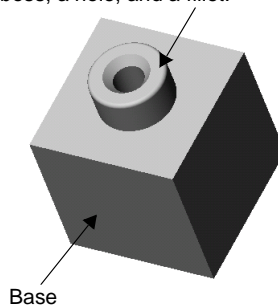
- 3 In the FeatureManager design tree, select the features you want to save as the library feature.

To select more than one feature, hold down the **Ctrl** key while you select. To select several features in a row, click the first feature, hold down the **Shift** key, then click the last feature.

*Do not* select the **Base** feature unless you plan to insert the library feature into an empty part.

- 4 Click **File, Save As**. The **Save As** dialog box appears.
- 5 From the **Save as type** list, select **Lib Feat Part Files (\*.sldlfp)**. Enter a name and click **Save**.

Library feature with three features: a boss, a hole, and a fillet.



The part icon beside the part name in the FeatureManager design tree changes to the library feature icon, and each feature that is included is marked with a letter **L**.

---

**NOTE:** To create a library feature from an existing part, open the part, select those features that you want to use in your library feature, and save them as a library feature part (\*.sldlfp).

---

The source part document may contain features that are not related in any way to the features that you selected. The software recognizes such cases, and gives you a choice when you save the library feature part. You can create either a simplified library feature part that does not include the unrelated features, or one that contains all the features. It is important to note that removing the unrelated features may cause rebuild errors in the source part, due to unresolved dependencies.

## Adding a Library Feature to a Part

There are two ways to add a library feature to a part:

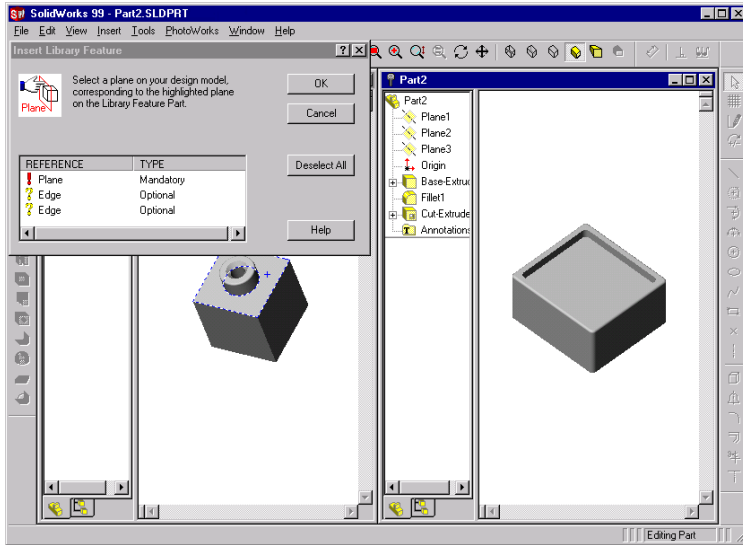
- Place the library feature in the Feature Palette (available for most, but not all types of features), then drag and drop the feature into the part. See **The Feature Palette Window** on page 11-6.
- Use **Insert, Library Feature** (for those types of features that are not supported in the Feature Palette).

### *To add a library feature to a part:*

- 1 With the target part open, click **Insert, Library Feature**. The **Insert Library Feature** dialog box appears.
- 2 Browse to the directory where the library feature is located, and select the library feature file (.sldlfp).
- 3 Click **Open**. Two windows and a dialog box appear, and tile automatically:
  - the library feature window
  - the target part window
  - the **Insert Library Feature** dialog box

In the **Insert Library Feature** dialog, there is at least one mandatory **Reference**, and there may be optional references also. A mandatory reference is preceded by an exclamation point; an optional reference is preceded by a question mark. Dimensional references are optional references.

- 4 To locate the library feature on the target part, click a **Reference** entity (**Plane**, **Edge**, **Face**, or **Vertex**) that is listed as **Mandatory** on the target part. The exclamation point in the **Reference** list changes to a check mark.

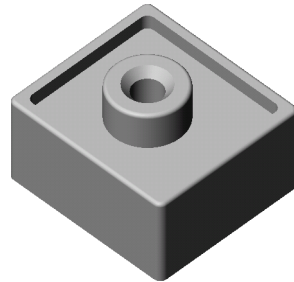


- 5 Select any optional references. As you click each item in the **Reference** list, notice that the corresponding entity is highlighted in the library feature window. Select the corresponding entity in the target part window.

To deselect an entity, either double-click the check mark or click **Deselect All**.

- 6 Click **OK**.

The library feature is added to the target part.



### Editing a Library Feature

You can change which features are included as part of a library feature in the library feature part document (.sldlfp):

- To add another feature, right-click the feature to add, and select **Add To Library**.
- To remove a feature, right-click the feature, and select **Remove From Library**.

After the library feature is inserted into a part, you can edit a library feature using the same techniques that you would use to edit any SolidWorks feature, such as **Edit Sketch** or **Edit Definition**.

You can move the library feature to a different location on the target part by modifying the locating dimensions. Double-click the library feature icon in the FeatureManager design tree to display the dimensions.

You can edit an existing library feature and save it with a new name (and .sldlfp extension) to create additional, similar library features.

## Adding Color

You can apply color to a library feature either in its own document or in a target part.

### *To change the color of library features:*

- 1 Click **Tools, Options**, and select the **Color** tab.
- 2 In the **Features** box, select **Library Feature**.
- 3 Click the **Edit** button and select a color from the color palette (or create a custom color), and click **OK**.
- 4 Select an option from the **Apply to** list to specify the scope of the changes:
  - **Active Document**. Apply the settings to *only* the currently active document.
  - **System Defaults**. Apply the settings to all *new* documents. The currently active document does not change.
  - **All Possible**. Apply the settings to both the **System Defaults** and the **Active Document**.
- 5 Click **OK**.

In a library feature document (**.sldlfp**), all the features that are included in the library feature use the new color.

In a target part, all inserted library features use the new color.

## Dissolving a Library Feature

After the library feature is inserted into a part, you can dissolve (break down) a library feature into the individual features it contains.

### *To break down a library feature into the features it contains:*

In the FeatureManager design tree, right-click the library feature icon, and select **Dissolve Library Feature** from the shortcut menu.

The library feature icon is removed, and each of the features it contains is listed individually in the FeatureManager design tree.

## The Feature Palette Window

---

The Feature Palette window gives you quick access to the features and parts you use most often. It helps you organize features and parts in a way that suits your specific needs.

Palette features and library features are similar in many ways. You create a palette feature in the same way as a library feature. The palette feature is displayed as a library feature in the FeatureManager design tree of the target part. You can dissolve a palette feature the same way as a library feature.

Palette features differ from library features in these ways:

- ❑ When you add a palette feature to a part, you can simply drag the feature roughly where you want it to be, then place it precisely by re-attaching and modifying any locating dimensions saved in the feature.
- ❑ You can edit the dimensions of the palette feature as you insert it into the part.
- ❑ Not all library features are suitable for use in the Feature Palette window. A palette feature is limited to *one* mandatory reference (one face, plane, edge, or vertex) on the target part.
- ❑ Library features that have multiple mandatory references (such as draft features that depend the selection of several faces) cannot be added through the Feature Palette window.

You can also use the Feature Palette window to add parts to an assembly. You can organize parts in logical groups, and you can see a miniature graphical view of each palette part. This makes it easy to locate, select, and insert the parts you want.

You can add fittings to piping assemblies by dragging them from the Feature Palette window. For more information, see Chapter 13, “Piping.”

The Feature Palette window is also used to apply forming tools to sheet metal parts. Forming tools are special parts that punch, stretch, or shape the sheet metal part, to create embossments, lances, flanges, louvers, ribs, and so on.

### Mandatory References

The limitation of a single mandatory reference does not mean that a palette feature can consist of only one feature. As long as the *parent* feature uses a single reference, the palette feature can include multiple *child* features. To examine the parent/child relationships, right-click the feature, and select **Parent/Child**.

You can determine if a library feature is suitable for use in the Feature Palette by inserting it as a library feature in a target part (see **Adding a Library Feature to a Part** on page 11-3). If there is only one mandatory reference listed, you can use the library feature as a palette feature.

## Feature Limitations

Because of the limitation of a single reference, lofts, sweeps, and shape features are allowed only in certain cases:


- For lofts, the single mandatory reference is the sketch plane of one of the profiles. This must be the face or plane where you drop the feature. Additional sketch planes must be offset from this plane, or from each other. The library feature must include the planes.
- For sweeps, the sweep *path* is the single mandatory reference. This must be a model edge. The sketch plane for the sweep *section* must be a **Perpendicular to Curve** plane, with the model edge as the curve (when you select the edge, click near the vertex where the plane should be placed). Include this plane in the library feature. Then, when you drop the palette feature in the target part, drop it on the edge that is the sweep path, near the appropriate vertex.
- For shape features, the shape must not be constrained to any entities. The definition can include only variations in **Pressure**, **Bend**, and **Stretch** (on the **Controls** tab of the **Shape Feature** dialog box).

## Displaying the Feature Palette Window

The Feature Palette window has several default folders. Each folder represents a directory; each palette item contained in that directory is displayed as a “thumbnail” graphic. Palette items can include parts and sheet metal forming tools (.sldprt files), and library features (.sldlfp files).

### ***To display the Feature Palette window:***

Click **Tools, Feature Palette**. The Feature Palette window stays open, and on top of the SolidWorks window, while you work. If the Feature Palette window is displayed when you exit SolidWorks, it will be displayed when you start SolidWorks the next time.





To dismiss the Feature Palette window, either click  in the upper-right corner of the window, or click **Tools, Feature Palette** again to remove the check mark from the item in the menu.

You can display the folder icons and the thumbnail graphics of palette items in two sizes. To change the size, right-click a blank area of the window, and select **View, Large Icons** or **Small Icons**.



## Navigating in the Feature Palette Window

When you display the Feature Palette window, it opens at the top level, or **Palette Home**. To open a folder, double-click it. These toolbar controls are similar to the controls in Internet Explorer.

-  **Go Backward**
-  **Go Forward**
-  **Reload**
-  **Home** (to **Palette Home**)
- The dropdown history list, with the names of the last folders opened (up to 16).

## Organizing Feature Palette Items

The Feature Palette window has several default folders. You can create and name new folders to suit your specific needs, and you can place items in each folder according to the way you want to organize the contents.

The SolidWorks software includes some palette items to get you started. They are stored in these folders by default:

*install\_directory\data\Palette Features\folder\_name*

*install\_directory\data\Palette Parts\folder\_name*

*install\_directory\data\Palette Forming Tools\folder\_name*

If you chose a different path for the **data** directories when you installed SolidWorks, these folders are placed in the path you specified instead.

You add individual items to a folder by placing parts, forming tools (**.sldprt**) or library feature parts (**.sldlfp**) in the appropriate folder.

A folder can contain individual palette items, and other folders. You can have as many folders, and as many levels of folders, as you want.


You can store palette items anywhere you want; the folders do not have to be located in the default path. You can have folders on different disk drives on your system, or on different computers on a network.

### ***To specify which folders appear at the Palette Home level:***

- 1 Click **Tools, Options, External References**.
- 2 From the **Show folders for** list, select the type of file for which you are defining a folder list: **Palette parts**, **Palette features**, or **Palette forming tools**.

The first time you edit the **Folders** list, the default paths for the palette folders are already listed. These folder lists are always used, whether the **Search document folder list for external references** check box is selected or not (this check box applies only to the **Documents** folder list).

You can have more than one folder in the **Palette Home** for each type of item.

- To add a new path to the list, click the **Add** button. In the **Choose Directory** dialog box, locate the path, and click **OK**.
  - To delete a path from the list, select the path in the **Folders** list, and click the **Delete** button.
  - To change the order of the list, select a path in the **Folders** list and use the **Move Up** or **Move Down** buttons to rearrange the list. Folders appear in the **Palette Home** in the order of the list.
- 3 Click **OK**. If you change the paths while the Feature Palette window is open, click **Reload**  to update the paths.

The icons in the Feature Palette window identify the type of item that the folder contains (with large icons only).

## Creating a Palette Item

There are several ways to add items to the Feature Palette.

- Move or copy existing parts, forming tools (.sldprt) or library feature parts (.sldlfp) into the appropriate folder, using Windows Explorer utilities.
- Drag the item from an open document and drop it into the Feature Palette window.
- Drag a document from Windows Explorer and drop it into the Feature Palette window.
- Drag a hyperlink to a palette item from the Internet Explorer (version 4.0 or later), and drop it into the Feature Palette window.

### ***To add a palette item from an open document:***

- 1 Create and save a new part, forming tool, or library feature part, or open an existing document (.sldprt or .sldlfp).
- 2 Display the Feature Palette window, and navigate to the folder where you want to store the palette item.
- 3 Drag the part from the graphics area, or the part icon from the top of the FeatureManager design tree of the active document, and drop it on the Feature Palette window. The **Save As** dialog box appears.
- 4 Either accept the current name, or enter a new name, and click **Save**.

### ***To add a palette item from Windows Explorer:***

- 1 Display the Feature Palette window, and navigate to the folder where you want to store the palette item.
- 2 In Windows Explorer, navigate to the folder that contains the palette item.
- 3 Drag the file from the Explorer window and drop it in the Feature Palette window.

**To add a palette item from a hyperlink in Internet Explorer:**

- 1 Display the Feature Palette window, and navigate to the folder where you want to store the palette item.
- 2 In Internet Explorer (version 4.0 or later), navigate to a location that contains hyperlinks to SolidWorks palette items.

For example, on the SolidWorks web site, in the Support section, navigate to the Solid Modeling Library page. You may also be able to obtain SolidWorks palette items from third-party vendors.

- 3 Drag a hyperlink from the Internet Explorer window, and drop it in the Feature Palette window. The document is saved in the specified folder.

You can also just *open* a hyperlinked item, without adding it to the palette. To open the document, drag the item and drop it into an empty area of the SolidWorks window. Then you can decide whether you want to add the item to the palette, or save it to a different folder, or close it without saving.

## Thumbnail Graphics

When you save a palette item, the thumbnail graphic reflects the view when the document is saved. Be sure to orient the part so that the thumbnail graphic looks the way you want it to look.


To make it clear which features are included in a palette feature, consider changing the color in the library feature part. Thumbnail graphics use a palette of 16 colors; if the color you choose for the feature is not available, the software replaces it with a similar color in the thumbnail graphic. See **Adding Color** on page 11-5.

---

**NOTE:** In the target part, palette features that consist of *more than one feature* use the color specified for **Library Features** under **Tools, Options, Color**.

Palette features that consist of *a single feature* are automatically dissolved when you add them. Therefore they use the color specified for the individual feature type.

---

Another way to emphasize the features is to select them, then click **Zoom to Selection**  before you save the library feature part. That way, the thumbnail graphic provides a close-up view of the included features.

## Working with Palette Items and Folders

### **To copy a palette item:**

Hold down the **Ctrl** key, then drag a palette item within the Feature Palette window. You can only copy an item this way in the same folder. The new item is named **Copy of <item>**.

### **To rename a palette item:**

You can rename the palette items and the folders, except the folders at the **Palette Home** level. Click-*pause*-click the name in the Feature Palette window, and enter a new name.

### **To edit a palette item:**


- 1 Right-click the item in the palette window, and select **Edit Palette Item**.  
– or –  
Drag the item from the Feature Palette window and drop it in an empty area of the SolidWorks window (not an empty area of a document window).  
The part (**.sldprt**) or library feature (**.sldlfp**) document is opened in a new window.
- 2 Edit the document as needed, then **Save**. The palette updates automatically to reflect the changes.

---

**NOTE:** The thumbnail graphic reflects the newly saved view. Be sure to orient the part so that the graphic looks the way you want it to look.

---

### **To change the contents of a palette folder:**

- 1 In Windows Explorer, navigate to the directory you want to modify.
- 2 Add, move, or delete the files as needed, using Windows file management commands.
- 3 Click **Reload**  on the toolbar in the Feature Palette window to update the folders.

### **To delete a palette item:**

Click the item in the Feature Palette window, and press **Delete**. The item is removed from the window, and the file is deleted from the corresponding folder.

### **To delete a palette folder:**

- 1 Exit SolidWorks.
- 2 Navigate to the directory to delete, move the contents to a new location, if necessary, then delete the directory.

## Adding Palette Items to Documents

### Adding a Palette Feature to a Part

Try using either **Hidden Lines Removed** or **Hidden in Gray** view mode, in order to see the preview and dimensions easily.

#### To add a palette feature to a part:

- 1 With a part open, click **Tools, Feature Palette**, and navigate to the folder that contains the feature.
- 2 Drag the feature from the Feature Palette window, observing the preview as you drag. Drop the feature in the general area of the face where you want the feature to be placed.

---



**NOTE:** You can also drag a hyperlink to a library feature part (.sldlfp) from Internet Explorer (4.0 or later) and drop it on the face of the part.

If you want to save the hyperlinked palette feature for future use, be sure to also drag a copy into the palette, or into an empty area of the SolidWorks window, then save it. Otherwise, no local copy of the document is saved.

Hyperlinked library feature parts have the same limitations and behaviors as library feature parts in the Feature Palette window.


---

The **Edit This Sketch** dialog box appears, and remains open while you position the sketch.

- 3 Click **Zoom to Selection**  to get a closer view of the feature.
- 4 Click **Modify Sketch** , and move or rotate the sketch as needed. See **Modify Sketch** on page 2-22 for more information.

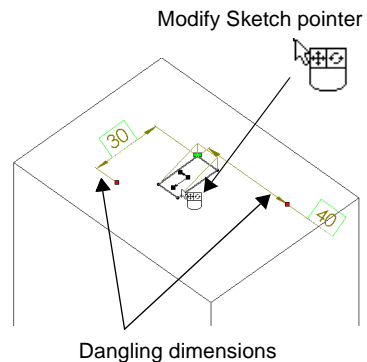
If the feature has locating dimensions, they are left dangling, and are displayed in brown.

- 5 Re-attach the dangling dimensions. Select a dimension, then drag the red handle on the dimension line.

The pointer has the  shape until it is over a suitable entity for re-attachment. When you release the handle over an edge or vertex, the dimension re-attaches and the handle turns green.

You can also add geometric relations to control the position of the sketch.

- 6 Adjust the values of the locating dimensions. (Double-click the dimension, change the value, and press **Enter**.) You can also adjust the locating dimensions in the next step, or later, by editing the appropriate sketch.
- 7 Click **Next** in the **Edit This Sketch** dialog box. The **Change Dimensions** dialog box appears.

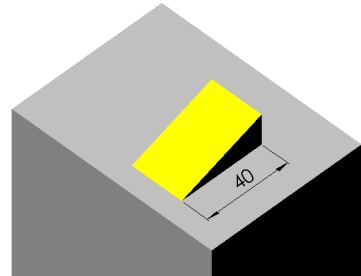


The **Name** and **Value** of each available dimension is displayed. If the dimensions are named in the part where you created the library feature, those names are used. Otherwise, the default dimension names **D1** through **Dn** are used. See **Dimension Properties** on page 2-36.

Some dimensions of the feature may be unavailable, because they are specified as **Internal Dimensions**. See **Controlling Access to Dimensions** on page 5-8 for more information.

- 8 To display a feature dimension on the model, click either the **Name** or **Value**.

To modify a dimension, double-click the **Value**, and enter a new value. Click **Apply** to see the changes.



- 9 When you are satisfied with the dimension values, click **Finish**.

If the palette item consists of more than one feature, a library feature icon appears in the FeatureManager design tree. If the palette item consists of a single feature, the item is automatically dissolved, and the appropriate icon for the feature type appears in the FeatureManager design tree. In either case, the name of the feature is the name of the item in the Feature Palette window.

## Adding a Palette Part to an Assembly

### *To add a palette part to an assembly:*

- 1 With the assembly open, click **Tools, Feature Palette**, and navigate to the folder that contains the desired part.
- 2 Drag the part from the Feature Palette window and drop it into the assembly window.

If the part has multiple configurations, the **Select a configuration** dialog box appears, with a list of the configurations in the part. Select the configuration you want, and click **OK**.

If you defined a **Mate Reference** in the part, you can create some types of mates automatically when you drop the part in the assembly. See **Defining a Mate Reference** on page 6-25.

For information about positioning and mating assembly components, see Chapter 6, “Working with Assemblies.”



If you are editing a piping sub-assembly, you can drag fittings (T-junctions, flanges, and so on) from the Feature Palette window and drop them on vertices in the sketch. See Chapter 13, “Piping.”

## Applying a Palette Forming Tool to a Sheet Metal Part

You can only apply forming tools through the Feature Palette window, and you can only apply forming tools to a part that has sheet metal features. You must have already inserted bends in the part before you attempt to apply a forming tool.

For information about creating sheet metal parts and forming tools, see Chapter 12, “Sheet Metal.”

### *To apply a palette forming tool to a sheet metal part:*

- 1 With a sheet metal part open, click **Tools, Feature Palette**, and navigate to the folder that contains the forming tools.
- 2 To apply forming tools to the sheet metal part in a flattened state, drag the rollback bar above the **Process-Bends** feature.
- 3 Drag the forming tool from the Feature Palette window to the face you want to deform. The face where you apply the forming tool corresponds to the stopping surface of the tool itself. By default, the tool travels downward. The material is deformed when the tool strikes the face where it is dropped. To toggle the direction of travel, and strike the opposite side of the material, press the **Tab** key. Observe the preview as you toggle the direction.
- 4 Drop the feature where you want it to be applied. If the forming tool includes an optional positioning sketch, that sketch is displayed when you drop the tool on the face.
- 5 Use **Dimension**  or **Modify Sketch**  to position the feature. As you add dimensions, the positioning sketch moves as a single entity. You cannot edit the dimensions of the forming tool itself. The absorbed sketch in the feature controls only the *location* of the feature, not its dimensions.
- 6 Click **Finish**. The forming tool is applied to the face, and the feature is added to the FeatureManager design tree, with the name of the forming tool.

## Palette Parts as Derived Parts

You can use the Feature Palette window to create a derived part, based on the selected palette part.

- 1 With an *empty* part document open, click **Tools, Feature Palette**, then navigate to the folder that contains the desired part.
- 2 Drag the part from the Feature Palette window and drop it into the part window.
- 3 When you are asked if you want to create a derived part:
  - Click **Yes** if you want to create a derived part (a part that has another part as its first feature). See **Derived Parts** on page 5-16 for more information.
  - Click **No** if you want to edit the palette part itself. The palette part is opened in the current window.

This chapter introduces the SolidWorks sheet metal functionality and describes:

- Designing sheet metal parts
- Rolling back and rebuilding a sheet metal design
- Creating sheet metal parts
- Adding and editing bends
- Selecting the type of bend allowance
- Adding and editing auto reliefs
- Adding features in the flattened state
- Creating hems
- Creating bends across multiple tabs
- Creating a flat pattern configuration
- Creating a sheet metal drawing
- Creating forming tools



## Designing Sheet Metal Parts

---

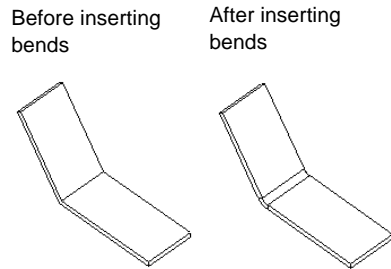
Sheet metal parts are generally used as enclosures for components or to provide support to other components.

You can design a sheet metal part on its own without any references to the parts it will enclose, or you can design the part in the context of the assembly containing the enclosed components. In the latter case, you can reference the enclosed components to create the geometry for the sheet metal part. For more information, see Chapter 6, “Working with Assemblies.”

### Creating Bends

You can create a solid model, then add bends to the model by specifying appropriate bend parameters such as bend radius and allowance.

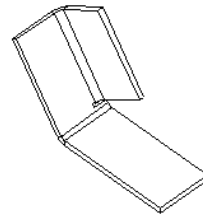
Adding bends to the model replaces the sharp corner with a filleted bend.



### Creating Bend Reliefs, Walls, and Tabs

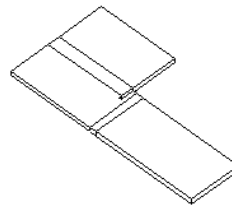
Walls or tabs can be added to a sheet metal part as designs dictate.

Relief cuts can be automatically added where necessary.



### Creating Flat Patterns

When the design is complete (or at any time during the design process) you can create a flat pattern of your sheet metal part.




## Bend Types

---


Three types of bends are available: sharp bends, round bends, and flat bends.

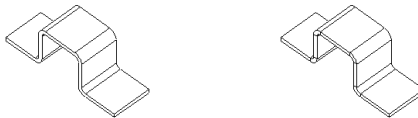
### Sharp Bends

A *sharp bend* is created by adding bends to a model that has sharp corners and a uniform thickness. Click the **Insert Bends**  tool on the Features toolbar, or click **Insert, Features, Sheet Metal, Bends** to add the bends. See **Using Sharp Bends** on page 12-10 for more information.



### Round Bends

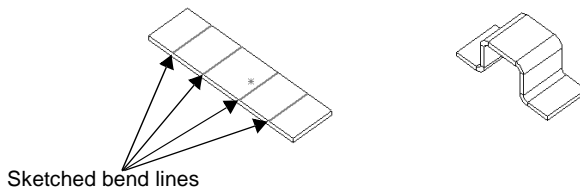
A *round bend* can be created from filleted corners created in the sketch, during the thin feature extrusion using the **Auto Fillet** option. When you click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**, the model does not change in appearance. However, bend lines and allowances are added to the part, and are visible when the model is flattened. See **Using Filleted Corners** on page 12-12 for more information.



Rounds bends are also created from cylindrical and conical faces. See **Using Cylindrical or Conical Faces** on page 12-13 for more information.


### Flat Bends

A *flat bend* is created from a line sketched in the flat state of a sheet metal part. Exiting the sketch creates the bend. See **Using a Flat Model** on page 12-16 for more information.



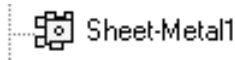
## Sheet Metal Features in the FeatureManager Design Tree

---

When you click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**, two distinct stages are applied to the sheet metal part. First, the part is flattened and a bend allowance is added. The developed length is calculated, based on the bend radius and bend allowance. Then, the flattened part is restored to the folded state to create the formed, folded version of the part. Three new features and icons appear on the FeatureManager design tree that are specific to sheet metal operations. These features are:

- Sheet-Metal**
- Flatten-Bends**
- Process-Bends**

These three features represent what can be thought of as a process plan for the sheet metal part.



Sheet-Metal1

**Sheet-Metal** contains the definition of the sheet metal part. This feature stores the default bend parameter information (thickness, bend radius, bend allowance, auto relief ratio, and fixed entity) for the entire part.

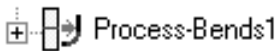


Flatten-Bends1

**Flatten-Bends** represents the flattened part. This feature contains information related to the conversion of sharp and filleted corners into bends.

Each bend generated from the model is listed as a separate feature under **Flatten-Bends**. Bends generated from filleted corners, cylindrical faces, and conical faces are listed as **Round-Bends**; bends generated from sharp corners are listed as **Sharp-Bends**.

The **Sharp-Sketch** listed under **Flatten-Bends** is the sketch that contains the bend lines of all sharp and round bends generated by the system. This sketch cannot be edited but can be hidden or shown.



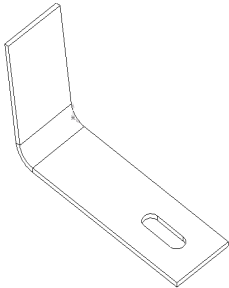
Process-Bends1

**Process-Bends** represents the transformation of the flattened part into the finished, formed part.

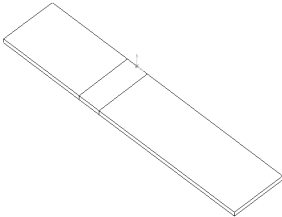
Bends created from bend lines specified in the flat are listed under this feature. **FlatSketch**, listed under **Process-Bends**, is a placeholder for these bend lines. This sketch can be edited, hidden, or shown.

Features that are listed after the **Process-Bends** icon in the FeatureManager design tree do not appear in the flattened view of the part. (You flatten the view of the part by rolling back the FeatureManager design tree to before the **Process-Bends** feature. See **Rolling Back and Rebuilding the Design** on page 12-6.)

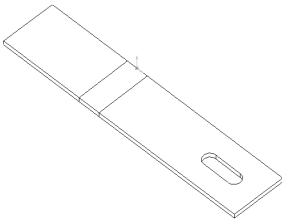
For example, in the sheet metal part shown here, the cut was added after the **Process-Bends** feature.



If you roll back to **Process-Bends**, the flattened part does not include the cut.



To see the cut in the flattened state, reorder the cut to occur before the **Process-Bends** feature.



## Rolling Back and Rebuilding the Design

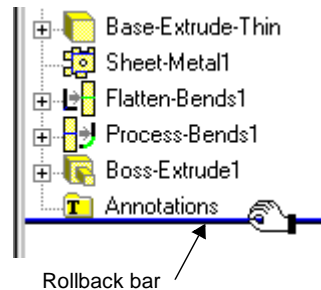
A *rollback* reverts the model to the state it was in before the selected feature was created. When you design sheet metal parts, you may roll back often to revert to previous stages in the design process. This section discusses the ways you can roll back the design.

### To roll back (Method 1):

- 1 In the FeatureManager design tree, select the feature you want to roll back to.
- 2 Click **Edit, Rollback**.

### To roll back by dragging the rollback bar (Method 2):

- 1 Place your pointer over the rollback bar in the FeatureManager design tree. The pointer changes to a hand.
- 2 Click to select the rollback bar. The bar changes color from yellow to blue.
- 3 Drag the rollback bar up the FeatureManager design tree until it is above the feature you want rolled back.



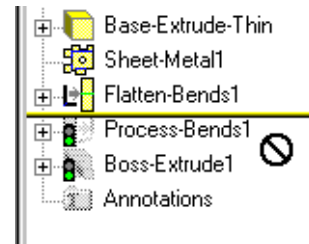

---

**NOTE:** You do not have to select a feature before you move the rollback bar. Also, the rolled-back icons are gray.



---

### To roll back to before Process-Bends (Method 3):

- 1 Click the **Flattened**  tool on the Features toolbar.
- 2 Click **Flattened**  again to restore the bends.



### To roll back to before Flatten-Bends (Method 4):

- 1 Click the **No Bends**  tool on the Features toolbar.
- 2 Click **No Bends**  again to restore the bends.

Throughout the following sections, whenever you need to roll back the design, you can use any of these methods.

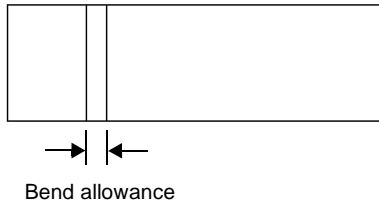
### To rebuild (roll forward):

To rebuild (roll forward) your part after rolling back the design, drag the rollback bar down the FeatureManager design tree. You can drag the rollback bar to the bottom of the FeatureManager design tree to rebuild the entire part, or drag it one or more features at a time to step through the regeneration of each rebuilt feature.

## Bend Allowance

---

Bend allowance represents the length of the bend region in the flattened state.



There are three ways to specify a sheet metal bend allowance. You can:

- Specify a k-factor
- Specify an explicit value
- Use a bend table

### K-factor

K-factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part. Bend allowance using a k-factor is calculated as follows:

$$BA = \pi(R + KT)A/180$$

where:

BA = bend allowance

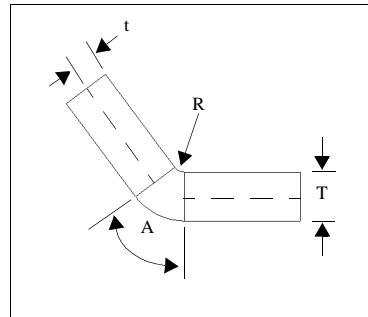
R = inside bend radius

K = k-factor, which is  $t / T$

T = material thickness

t = distance from inside face to neutral sheet

A = bend angle in degrees



### Explicit Bend Allowance Value

You can specify an explicit bend allowance for any sheet metal bend by entering the value when creating the bend.

## Bend Table

A sample bend table is provided in *install\_directory\lang\english\Sample.btl*. To use your own bend table when performing sheet metal operations, copy and rename this file and edit it to specify required bend allowances.

---

**NOTE:** All lengths must be specified in meters.

---

You can edit the bend table using any text editor. A portion of the sample bend table is shown next.

```

sample.btl - Notepad
File Edit Search Help
-----
Bend Allowance Tables
-----
Version: 1
Material: Steel
Units: meters

Thickness: 0.0005
Bend Radius (read across)      0.0000  0.0005  0.0010  0.0015  0.0020  0.0025  0.0030
Opening Angle (read down)
    5      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   10      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   20      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   30      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   45      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   50      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   70      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   80      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
   90      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  100      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  110      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  120      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  130      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  140      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  145      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  160      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  170      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
  180      0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002

Thickness: 0.0010
Bend Radius (read across)      0.0000  0.0005  0.0010  0.0015  0.0020  0.0025  0.0030

```

---

**NOTE:** The sample bend table is provided only for informational purposes. The values in this table do not represent any actual bend allowance values.

---

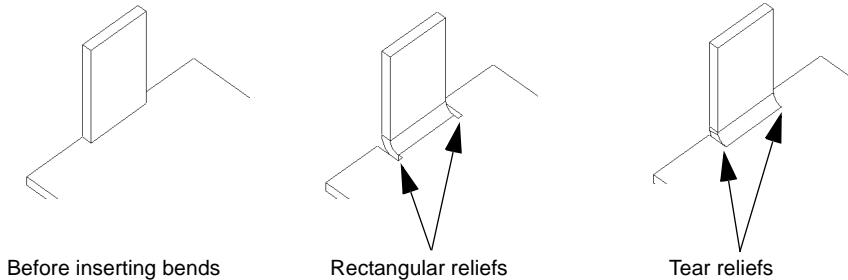
If the thickness of the part or bend angle falls between values in the table, the software interpolates the values to calculate the bend allowance.

## Auto Reliefs

---

You can automatically add relief cuts at bend regions when inserting bends. The SolidWorks software supports two types of reliefs:

- **Rectangular**
- **Tear**



If only one relief cut is needed, the software adds only one relief cut. If you select to automatically add **Rectangular** reliefs, you must specify the **Relief ratio**.

### Relief Ratio

The **Relief ratio** represents the width of the **Rectangular** relief cut and the depth by which the side of the **Rectangular** relief cut extends past a bend region, expressed as a ratio of material thickness.

$$d = (\text{relief ratio}) * (\text{part thickness})$$

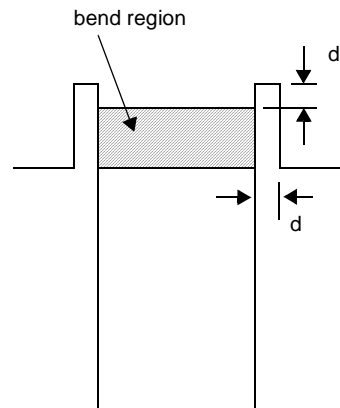
---

**NOTE:** **Relief ratio** must have a value between 0.05 and 2.0.

---

The higher the value of the relief ratio, the larger the size of the relief cut automatically added during insertion of bends.

You can edit a **Rectangular** relief cut to specify different width and depth dimensions. See **Editing Auto Reliefs** on page 12-22 for more information.






## Creating a Sheet Metal Part

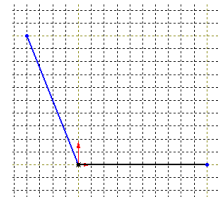
You create sheet metal parts using sharp bends, filleted corners, cylindrical faces, conical faces, rips, and flat models.

### Using Sharp Bends

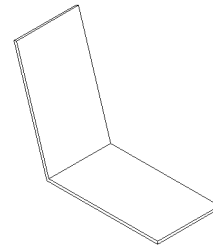
The following example demonstrates the creation of a sheet metal part based on a thin feature base. You can create a sheet metal part from any part that has a uniform thickness, and a thin feature extrusion is a quick way to create such a base.

#### To create a thin-feature base:



- 1 Create a new part, open a sketch, and sketch an open profile for the base.
- 2 Click **Extruded Boss/Base**  or **Insert, Base, Extrude**.
- 3 In the **Extrude Feature** dialog box, specify the desired **Depth** and **Type**.
- 4 On the **Thin Feature** tab, specify the desired **Type** and **Wall Thickness**.
- 5 Click **OK**.

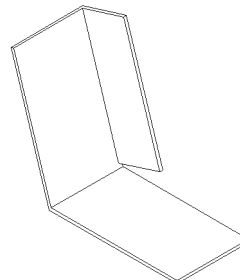
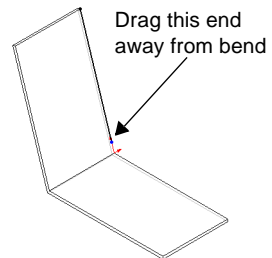


For more information about thin feature extrusions, see **Thin Features** on page 4-6.



#### To add a wall:

- 1 Open a sketch on the face of the part where the new wall will be attached.
- 2 Select the edge of the model and click **Convert Entities**  or **Tools, Sketch Tools, Convert Entities**.
- 3 Drag the vertex nearest to the bend a small distance away from the bend to allow for the bend radius.
- 4 Click **Extruded Boss/Base**  or **Insert, Boss, Extrude**.
- 5 On the **End Condition** tab, set the **Type** to **Blind** and specify the **Depth**.
- 6 On the **Thin Feature** tab, specify the same **Wall Thickness** as the base feature, and, if necessary, select the **Reverse** check box to extrude the wall in the opposite direction.
- 7 Click **OK**.



The new wall is added. It is generally a good practice to add all the walls before inserting the bends.

**To insert the bends:**

- 1 Click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**.

The **Flatten-Bends** dialog box appears.

To insert bends, you must select a *fixed face*. The fixed face remains in place when flattening the part during the creation of the **Flatten-Bends** feature.

- 2 In the model, select the fixed face.

The name of the fixed face appears in the **Fixed edge or face** box.

- 3 Enter a value in the **Default bend radius** box.

- 4 Select a **Bend allowance** type and value:


- If you select **Use bend table**, select the name of the bend table file. See **Bend Table** on page 12-8 for more information.
- If you select **Use k-factor**, specify the value based on the calculation. See **K-factor** on page 12-7 for more information.
- If you select **Use bend allowance**, specify the value.

- 5 To automatically add auto reliefs wherever needed, make sure that the **Use auto relief** check box is selected, then choose the desired type of auto relief cut.

If you choose **Rectangular**, then you must specify the **Relief ratio**. See **Relief Ratio** on page 12-9 for more information.

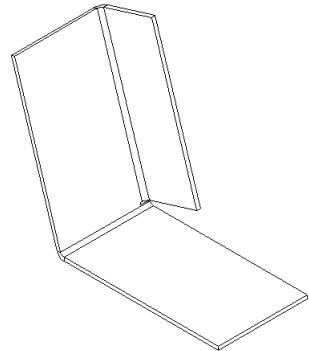
- 6 Click **OK**.

The sharp corners of the part are replaced by bends, and the necessary relief cuts are added.

- 7 Double-click the **Flatten-Bends** feature (or click the  beside its name) to expand the feature.

The **Sharp-Bend** features and a **Sharp-Sketch** feature are listed under the **Flatten-Bends** feature.

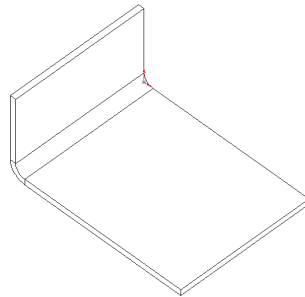
To change the fixed face, right-click the **Sheet-Metal** feature in the FeatureManager design tree, and select **Edit Definition**. Select a different planar face on the model, and click **OK**.




## Using Filleted Corners

**To create a sheet metal part using filleted corners:**

- 1 Create a thin feature part:
  - a) Select the **Auto fillet** check box in the **Thin Feature** tab of the **Extrude Thin Feature** dialog box.
  - b) Specify the **Fillet Radius** (inner bend radius).
  - c) Click **OK**.



- 2 Click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**.

The **Flatten-Bends** dialog box appears.

To insert bends, you must select a *fixed face*. The fixed face remains in place when flattening the part during the creation of the **Flatten-Bends** feature.

- 3 On the model, select the fixed face.

The name of the fixed face appears in the **Fixed edge or face** box.

- 4 Set the **Default bend radius** to zero (**0**).

- 5 Specify a **Bend allowance** method and value:

- If you select **Use bend table**, select the name of the bend table file. See **Bend Table** on page 12-8 for more information.
- If you select **Use k-factor**, specify the value based on the calculation. See **K-factor** on page 12-7 for more information.
- If you select **Use bend allowance**, specify the value.

- 6 Click **OK**.

A **RoundBend** feature appears in the FeatureManager design tree under the **Flatten-Bends** feature.

---

**NOTE:** You can also create a sheet metal part with round bends by extruding a thin feature part created from a sketch containing sketch fillets or tangent arcs.


---

To change the fixed face, right-click the **Sheet-Metal** feature in the FeatureManager design tree, and select **Edit Definition**. Select a different planar face on the model, and click **OK**.

## Using Cylindrical or Conical Faces

### To create a sheet metal part with cylindrical or conical faces:

- 1 Create a thin feature part with one or more cylindrical or conical faces:
  - At least one end face of any cylindrical or conical face must have at least one linear edge.
  - Any adjacent planar and cylindrical or conical faces must be tangent.

- 2 Click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**.

The **Flatten-Bends** dialog box appears.

To insert bends, you must select a *fixed edge*. The fixed edge remains in place when flattening the part during the creation of the **Flatten-Bends** feature.

- 3 On the model, select a linear edge on an end face of a cylindrical or conical face as the fixed edge.

The name of the fixed edge appears in the **Fixed edge or face** box.

- 4 Set the **Default bend radius** to zero (0).

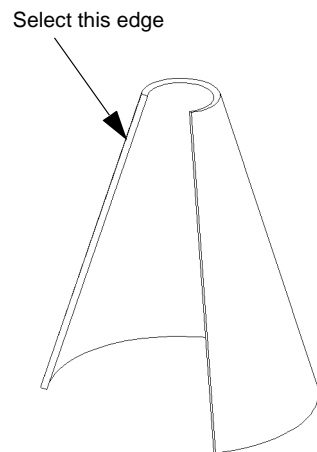
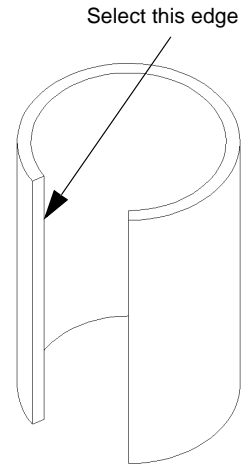
- 5 Specify a **Bend allowance** method and value:

- If you select **Use bend table**, select the name of the bend table file. See **Bend Table** on page 12-8 for more information.
- If you select **Use k-factor**, specify the value based on the calculation. See **K-factor** on page 12-7 for more information.
- If you select **Use bend allowance**, specify the value.

- 6 Click **OK**.

A **RoundBend** feature appears in the FeatureManager design tree under the **Flatten-Bends** feature.

To change the fixed edge, right-click the **Sheet-Metal** feature in the FeatureManager design tree, and select **Edit Definition**. Select a different edge on the model, and click **OK**.



---

**NOTE:** You cannot add bosses or apply forming tools to cylindrical or conical faces on sheet metal parts.


---

## Using Rips

You can rip the inside linear edges of adjacent planar faces in a sheet metal part so that you can flatten the part.

### **To create a sheet metal part using rips:**

- 1 Create a part of uniform thickness with adjacent planar faces that form one or more linear edges or a chain of linear edges. Chained edges must be adjacent to a common planar face.

- 2 Click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**.

The **Flatten-Bends** dialog box appears.

To insert bends, you must select a *fixed face*. The fixed face remains in place when flattening the part during the creation of the **Flatten-Bends** feature.

- 3 On the model, select the fixed face.

The name of the fixed face appears in the **Fixed edge or face** box.

- 4 Enter a value in the **Default bend radius** box.

- 5 Select a **Bend allowance** type and value:

- If you select **Use bend table**, select the name of the bend table file. See **Bend Table** on page 12-8 for more information.
- If you select **Use k-factor**, specify the value based on the calculation. See **K-factor** on page 12-7 for more information.
- If you select **Use bend allowance**, specify the value.

- 6 To automatically add auto reliefs wherever needed, make sure that the **Use auto relief** check box is selected, then choose the desired type of auto relief cut.

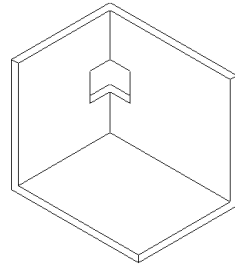
If you choose **Rectangular**, then you must specify the **Relief ratio**. See **Relief Ratio** on page 12-9 for more information.

- 7 Click the **Rip** tab.

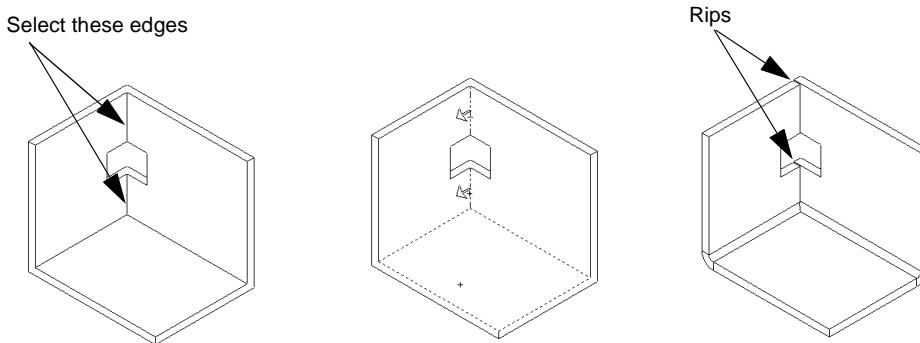
- 8 On the model, select the *inside linear edges* to rip.

Note the arrow direction of each selected edge. To create the rip in the opposite direction, click the name of each edge in the dialog box, and click **Change direction**. All of the arrows must point in the same direction

- 9 Click **OK**.

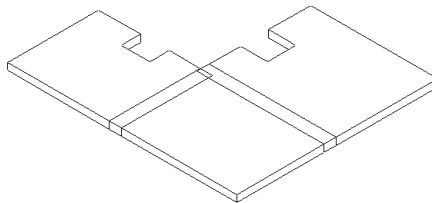


If an informational message stating that auto relief cuts were made for one or more bends is displayed, click **OK**.



The **Rip** feature appears under the **Sheet-Metal** feature. The **Sharp-Bend** features and a **Sharp-Sketch** feature are listed under the **Flatten-Bends** feature.


The following illustration shows the sheet metal part in the flattened state.



To change the direction of a rip, right-click the **Rip** feature in the FeatureManager design tree, and select **Edit Definition**. Click the name of each edge, select **Change direction**, and click **OK**.

To change the fixed face, right-click the **Sheet-Metal** feature in the FeatureManager design tree, and select **Edit Definition**. Select a different planar face on the model, and click **OK**.

---

**NOTE:** You can create rips in any part before inserting bends to create a sheet metal part. Create a part according to Step 1, click **Insert Rip**  on the Features toolbar, or click **Insert, Features, Sheet Metal, Rip**, and create the rips. Then insert the bends to create the sheet metal part.

---

## Using a Flat Model

While you should design sheet metal parts in the bent-up state (using either sharp corners or filleted corners), you can also model your parts from a flat sheet. You need to know the developed length required for your part and the locations of all bend lines in the part.

### To create a sheet metal part from a flat model:

- 1 Create a flat part. You can use either a solid or thin feature extrusion. The model must be of uniform thickness.

- 2 Click **Insert Bends**  or **Insert, Features, Sheet Metal, Bends**.

The **Flatten-Bends** dialog box appears.

To insert bends, you must select a *fixed face*. The fixed face remains in place when flattening the part during the creation of the **Flatten-Bends** feature.

- 3 On the model, select the fixed face.

The name of the fixed face appears in the **Fixed edge or face** box.

- 4 Enter a value in the **Default bend radius** box.

- 5 Select a **Bend allowance** type and value:

- If you select **Use bend table**, select the name of the bend table file. See **Bend Table** on page 12-8 for more information.
- If you select **Use k-factor**, specify the value based on the calculation. See **K-factor** on page 12-7 for more information.
- If you select **Use bend allowance**, specify the value.

- 6 Click **OK**.

An informational message is displayed stating that no bends were found. Click **OK**.

The sheet metal features appear in the FeatureManager design tree even though the part appears unchanged.

- 7 Expand the **Process-Bends** feature in the FeatureManager design tree.

- 8 Right-click the **Flat-Sketch** feature under **Process-Bends** and select **Edit Sketch**.

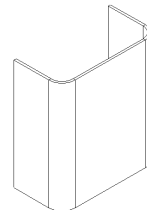
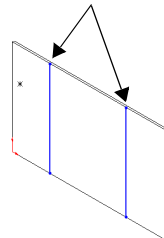
- 9 Sketch lines where you want bend lines.

- 10 To change the fixed face, place a sketch point on a different face.

- 11 Exit the sketch to rebuild the part.

Bends are created using the sketched bend lines and the specified bend parameter information.

Sketched bend lines




---

**NOTE:** You cannot specify a bend line for a flat bend that lies between existing sharp or round bends.




---

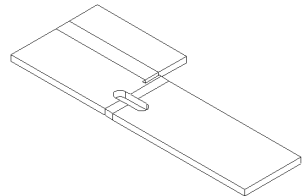
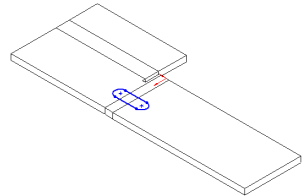
## Adding Features in the Flattened State

---




When designing sheet metal parts, it may be necessary to work in the flattened state to add features such as cuts across bend lines and tabs. You can roll back to the flattened state of a sheet metal part by clicking the **Process-Bends** feature and clicking **Edit, Rollback**, or by dragging the rollback bar in the FeatureManager design tree.

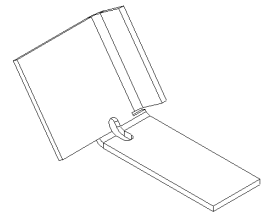
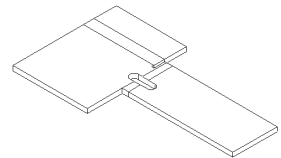
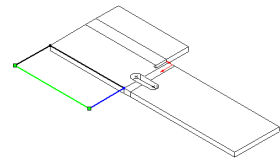
### *To create a cut across a bend line:*

- 1 Click **Flattened**  to roll back above the **Process-Bends** feature. (The part is shown in the flattened state.)
- 2 Select the top face of one of the flat regions and open a sketch.
- 3 Sketch the profile of the cut.
- 4 Click **Extruded Cut**  or **Insert, Cut, Extrude**.
- 5 Set **Type** to **Through All**, and click **OK**.
- 6 Click **Flattened**  again to rebuild the part.  
The part is folded back with the cut.



### *To create a tab:*

- 1 Click **Flattened**  to roll back above the **Process-Bends** feature. (The part is shown in the flattened state.)
- 2 Select the top face of one of the flat regions and open a sketch.
- 3 Sketch a tab with one side coincident with a model edge.
- 4 Click **Extruded Boss/Base**  or **Insert, Boss, Extrude**.
- 5 In the **Extrude Feature** dialog box:
  - Select the **Link to Thickness** check box to set the boss thickness equal to the base thickness.
  - Select the **Reverse Direction** check box if the extrusion is extending the wrong way.
- 6 Click **OK**.
- 7 Click **Flattened**  again to rebuild the part.  
The part is folded back with the tab.





## Adding Bends to a Flattened Model


---

You can create hems and bends across multiple tabs using a flat model.

### Hems

You can make hems of various types in sheet metal parts. The SolidWorks software supports open, closed, double, and tear-drop hems.

#### *To create a hem:*

- 1 In an existing sheet metal part, click the  beside a **Process-Bends** feature.
- 2 Right-click the **Flat-Sketch** feature and select **Edit Sketch**.
- 3 Sketch a line to define the fold of the hem, and exit the sketch.

The part folds on the sketched line, using the default radius and angle. You need to edit the bend to give it radius and angle values suitable for a hem.

- 4 Right-click the **FlatBend** feature and select **Edit Definition**.
- 5 To enter a new value for the bend radius, click to clear the **Use Default** check box, then type the value in the **Radius** box.

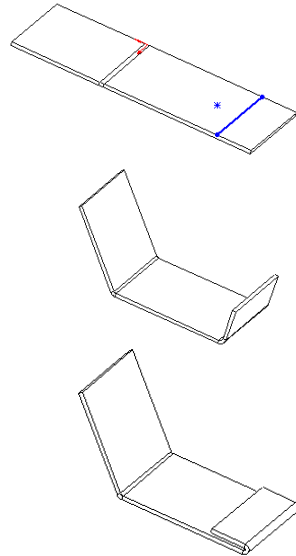
---

**NOTE:** The bend radius should allow for a small gap between overlapping faces so that they do not touch.

---

- 6 Enter a new value for the bend angle in the **Angle** box.
- 7 To reverse the direction of the bend, select the **Bend Down** check box.
- 8 Click **OK**.


The hem returns to the folded state using the new values.



## Bends Across Multiple Tabs

You can sketch a single bend line to create bends across multiple tabs.

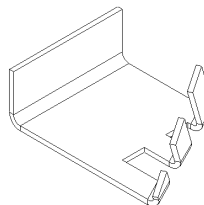
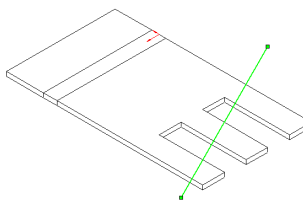
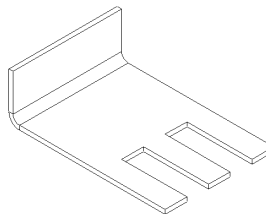
### **To create a bend across multiple tabs:**

- 1 In an existing sheet metal part that has multiple tabs, click the  beside a **Process-Bend** feature.
- 2 Right-click the **Flat-Sketch** feature and select **Edit Sketch**.
- 3 Sketch a line to define the bend across the tabs, and exit the sketch.

The part bends on the sketched line, using the default radius and angle. You can edit the bend to give it different radius and angle values.

- 4 Right-click the **FlatBend** feature and select **Edit Definition**.
- 5 To enter a new value for the bend radius, click to clear the **Use Default** check box, then type the value in the **Radius** box.
- 6 Enter a new value for the bend angle in the **Angle** box.
- 7 To reverse the direction of the bend, select the **Bend Down** check box.
- 8 Click **OK**.

The tabs return to the folded state using the new values.



## Editing Bends

---

You can edit bend information for sheet metal features. The scope of the changes depends upon which feature is edited.

### Editing Bend Parameters

*To edit bend parameters for the entire part:*

- 1 Right-click the **Sheet-Metal** feature in the FeatureManager design tree and select **Edit Definition**.
- 2 In the **Sheet-Metal** dialog box:
  - Change the value in the **Default bend radius** box.
  - In the **Bend allowance** section, change the method.
- 3 Click **OK**.

*To edit bend parameters for all bends listed under Flatten-Bends:*

- 1 In the FeatureManager design tree, right-click the **Flatten-Bends** feature and select **Edit Definition**.
- 2 In the **Flatten-Bends** dialog box:
  - Click to clear the **Use default** check box, then change the value in the **Default bend radius** box.
  - In the **Bend allowance** section, click to clear the **Use default** check box, then change the method.
- 3 Click **OK**.

*To edit bend parameters for all bends listed under Process-Bends:*

- 1 In the FeatureManager design tree, right-click the **Process-Bends** feature and select **Edit Definition**.
- 2 In the **Process-Bends** dialog box:
  - Click to clear the **Use default** check box, then change the value in the **Default bend radius** box.
  - In the **Bend allowance** section, click to clear the **Use default** check box, then change the method.
- 3 Click **OK**.

**To edit bend parameters for an individual bend under Flatten-Bends:**

- 1 In the FeatureManager design tree, click the  beside the **Flatten-Bends** feature.
- 2 Right-click the bend you want to edit and select **Edit Definition**.
- 3 On the **Bend** tab:
  - Click to clear the **Use Default** check box, then change the value in the **Radius** box.
  - Change the value in the **Angle** box.
  - Change the value in the **Order** box to change the order of the bends.

---

**NOTE:** This value specifies the order in which the bends are made in the final fabrication of the part. The value is not used by the software; it is for your documentation purposes only.

---

- In the **Bend Allowance** section, click to clear the **Use Default** check box, then change the method.
- 4 Click **OK**.

**Notes on editing bends:**

- You cannot change the bend angle for sharp bends. This angle and the bend radius are calculated from the model.
- You cannot change the bend angle or radius for round bends. The bend angle is calculated from the model.

**To edit bend parameters for an individual bend under Process-Bends:**

- 1 In the FeatureManager design tree, click the  beside the **Process-Bends** feature.
- 2 Right-click the bend you want to edit and select **Edit Definition**.
- 3 On the **Bend** tab:
  - Click to clear the **Use Default** check box, then change the value in the **Radius** box.
  - Change the value in the **Angle** box.
  - To reverse the direction of the bend, select the **Bend Down** check box.
  - Change the value in the **Order** box to change the order of the bends.

---

**NOTE:** This value specifies the order in which the bends are made in the final fabrication of the part. The value is not used by the software; it is for your documentation purposes only.

---

- In the **Bend Allowance** section, click to clear the **Use Default** check box, then change the method.
- 4 Click **OK**.

## Editing Auto Reliefs

You can edit the relief cuts automatically added to the bend regions in sheet metal parts.

---

**NOTE:** You cannot edit relief cuts in sheet metal parts created in versions prior to SolidWorks 99.

---

### *To change the type of relief cut for all bends:*

- 1 Right-click the **Sheet-Metal** feature in the FeatureManager design tree and select **Edit Definition**.
- 2 In the **Auto Relief** section, change the type of relief cut.  
If you change the type of relief cut to **Rectangular**, you must specify the **Relief ratio**.
- 3 Click **OK**.

### *To change the type and size of a relief cut for an individual bend:*

- 1 In the FeatureManager design tree, click the  beside the **Flatten-Bends** feature.
- 2 Right-click the bend you want to edit and select **Edit Definition**.
- 3 Click the **Auto Relief** tab.
- 4 Click to clear the **Use default relief** check box.
- 5 Change the type of relief cut in the **Relief Type** section, if desired.
- 6 If the type of relief cut is **Rectangular**, change the **Width** and **Depth** values, if desired.
- 7 Click **OK**.

## Creating a Flat Pattern Configuration

---

Creating a flat pattern is equivalent to suppressing the last **Process-Bends** feature and all features after it. You can save this as a configuration to use in a drawing.

### *To create a flat pattern configuration:*

- 1 Create and name a new configuration. (For information about creating a configuration, see online help and Chapter 8, “Working with Configurations.”)
- 2 Select the last **Process-Bends** feature in the model and all features after it.
- 3 Click **Edit, Suppress**. Also, you can right-click the **Process-Bends** feature, select **Properties**, and select the **Suppressed** check box.
- 4 Save the part.

You should create the flat pattern configuration after the entire design of the part is completed. This ensures that all features are shown in the flat pattern.

## Creating Drawings of Sheet Metal Parts

---

You can create drawings of sheet metal parts. For more information on creating drawings, see online help and Chapter 9, “Drawings.”

### *To create a drawing of a flat pattern:*

- 1 Create a part configuration showing a flat pattern as presented in the previous section. If none of the standard views shows the flat part in such a way that the bend lines are clearly displayed, create a named view in a suitable orientation.
- 2 Create a new drawing.
- 3 Click **Insert, Drawing View, Named View**.
- 4 Change to the part window, click anywhere, then select the desired view from the **Drawing View - Named View** dialog box.
- 5 Return to the drawing window and click to place the view on the sheet.


### *To remove the display of the bend region lines:*

- 1 In the drawing window, right-click the drawing view in the FeatureManager design tree.
- 2 Select **Tangent Edge, Tangent Edges Removed**.

### *To show the bend lines in the view of the flat pattern:*

- 1 Either in the part document or the drawing view, expand the **Flatten-Bends** and **Process-Bends** features.
- 2 Right-click **Sharp-Sketch** and **Flat-Sketch** in the FeatureManager design tree, and click **Show**.

If the drawing includes views of the model in both the flattened and folded states, you can show the bend lines in the view of the flat pattern only. First show the bend lines (see previous section), then convert the bend lines in the drawing view of the flat pattern.

- 1 Select the bend lines in the drawing view of the flat pattern.
- 2 Click the **Convert Entities**  tool on the Sketch Tools toolbar with the view of the flat pattern active.
- 3 Repeat Steps 1 and 2 from the previous section, but click **Hide**.
- 4 To change the line style of the converted bend lines, click **Line Style** on the Line Format toolbar and select the desired line style.

### *To change the configuration in the drawing view:*

- 1 Right-click the drawing view in which you want to change the configuration, and select **Properties**.
- 2 In the **Use named configuration** box, select a different configuration.
- 3 Right-click the drawing view again, and select **Update View**, if necessary.

## Tips for Designing Sheet Metal Parts

---

The order in which you add features is key to getting the desired results. The recommended approach for designing sheet metal parts is as follows:

- Design the part in the bent-up state, if possible.
- Add **Insert, Features, Sheet Metal, Bends** only after the functional design is complete; that is, add all the features before inserting bends.
- If some features such as bend reliefs and additional walls need to be added after inserting bends, add these features after rolling the model back to the **Process-Bends** feature. This step flattens the model.
- If you do not want certain features to be visible in the flattened state (features such as holes that are drilled after the sheet metal part is bent), add these features *after* the **Process-Bends** feature.
- To ensure that the part has uniform thickness, use the **Link to Thickness** check box in the **Extrude** dialog box.

## Forming Tools

---

Forming tools act as dies that bend, stretch, or otherwise form sheet metal. You apply forming tools to sheet metal parts to create such form features as louvers, lances, and ribs.

Create forming tools using bosses, cuts, fillets, chamfers, and so on, and save forming tools as SolidWorks parts on the Feature Palette. You can only use forming tools from the Feature Palette, and you can only apply forming tools to sheet metal parts. For more information about using forming tools, see **Applying Forming Tools** on page 12-27.

The SolidWorks software includes some sample forming tools to get you started. They are stored in this directory:

*install\_directory\data\Palette Forming Tools\folder\_name*

Examine the sample forming tools before creating your own. You may find that by editing a sample forming tool, you can create a forming tool that meets your needs. For more information about editing an existing forming tool in the Feature Palette, see **Working with Palette Items and Folders** on page 11-11.

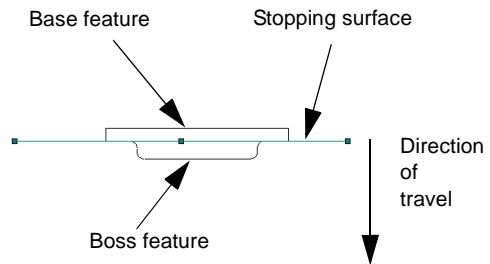
## Creating Forming Tools

### ***To create a forming tool:***

- 1 Open a new part document, then open a sketch on the horizontal plane. By default, the horizontal plane is **Plane2**.
- 2 To create the base feature, sketch a profile, and extrude the profile *above* the horizontal plane.

- 3 To create the boss feature, select the bottom face of the base feature, or select the horizontal plane, and open a sketch.

The horizontal plane is the *stopping surface* of the forming tool. To strike and deform sheet metal using a forming tool, the direction of travel of the forming tool must be along the vertical axis. Thus, the entire forming tool, except the base on which you create it, must lie below the stopping surface.



- 4 Sketch a profile for the boss centered around the origin.

The origin determines the drop point of the forming tool when applying the forming tool to a sheet metal part.

- 5 Dimension the profile, then extrude the profile *below* the stopping surface. The boss can be extruded or revolved.

To shape the forming tool, add features such as cuts, fillets, chamfers, and so on, to the boss.

---

**NOTE:** The minimum radius of curvature of the forming tool must be greater than the thickness of the sheet metal part to which it will be applied. To find out the minimum radius of curvature of the forming tool, use **Tools, Check**. For more information, see **Force Regeneration with Verification on Rebuild** on page 5-19.



---

- 6 Remove the base feature of the forming tool.

When you apply the forming tool to a sheet metal part, the stopping surface of the forming tool corresponds to the sheet metal face to which it is applied.



---

**TIP:** To remove the base feature of the forming tool, you can select the bottom face of the base feature, and open a sketch. Project the face onto the sketch plane by clicking **Convert Entities** . Click **Extruded Cut** , and add a **Through All** cut.

---



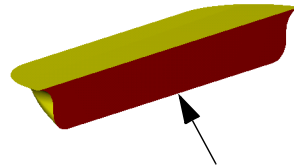
**To create an opening on a forming tool:**

Some types of form features, such as louvers and lances, create openings on sheet metal parts. To indicate which forming tool faces create openings, change the color of these faces to *red*.

- 1 Right-click the face designed to create an opening, and select **Face Properties**.
- 2 In the **Entity Property** dialog box, click **Change Color**.
- 3 Select the red box in the first column in the second row, then click **OK**.

In the **Entity Color** section, **Red** should be 255; **Green** and **Blue** should be 0.

- 4 Click **OK** again to close the **Entity Property** dialog box.



Make this face red to indicate an opening on the sheet metal part

**Creating Positioning Sketches**

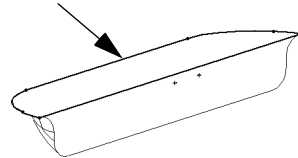
When you apply a forming tool to a sheet metal part, you cannot change the dimensions of the forming tool. You can, however, edit the position of the forming tool on the sheet metal face.

You can create a *positioning sketch* on the top face of the forming tool. When you apply the forming tool to the sheet metal part, you use the positioning sketch to locate the form feature on the face of the sheet metal part.


**To create a positioning sketch:**

- 1 Select the top face of the boss feature, and open a sketch.
- 2 Create a profile to locate the form feature on the sheet metal part. Close the sketch, and rename it, if desired.

Positioning sketch




---

**TIP:** To easily create a profile to locate the form feature, you can select the top face of the forming tool and open a sketch. Project the face onto the sketch plane by clicking **Convert Entities** , then close the sketch.

---

- 3 To hide the positioning sketch so that it does not appear in the thumbnail graphic of the forming tool displayed in the Feature Palette, right-click the sketch in the FeatureManager design tree, and select **Hide**.

## Controlling Access to Dimensions

You can prevent other users from changing certain dimensions of the forming tool by controlling access to its dimensions. You can also use this function to reduce the clutter of unnecessary dimensions in the display.

See **Controlling Access to Dimensions** on page 5-8 for more information.

## Saving and Adding Forming Tools to the Feature Palette

- 1 Save the part document (**.sldprt**).
- 2 Add the forming tool to the Feature Palette using one of the following methods:
  - Move or copy the forming tool part document to the desired Palette forming tools directory using Windows Explorer utilities. The default Palette forming tools directory path is *install\_directory\data\Palette Forming Tools\folder\_name*.
  - Click **Tools, Feature Palette**, then navigate to the desired forming tools folder. Drag the part icon from the FeatureManager design tree of the forming tool part, and drop it on the Feature Palette window.

If you want to store the forming tool in a different location, you must specify the directory path for the Feature Palette to use. For more information, see **Organizing Feature Palette Items** on page 11-8.

## Applying Forming Tools

You can only apply forming tools to a part that has sheet metal features; you must have already inserted bends in the part before you attempt to apply a forming tool.

Apply the forming tool to the sheet metal part by dragging the forming tool from the Feature Palette window and dropping it on a sheet metal face. By default, the forming tool deforms the opposite face of the material to which it is applied. To deform the face to which the forming tool is applied, press the **Tab** key as you drag. When you press the **Tab** key, the preview updates automatically.

A form feature is added to the FeatureManager design tree. Locate the form feature on the sheet metal part by using its positioning sketch. For more information, see **Applying a Palette Forming Tool to a Sheet Metal Part** on page 11-14.

You can create patterns of form features. See **Pattern and Mirror** on page 4-40 for more information.

You can create a special type of sub-assembly that builds a routing path between components. A routing sub-assembly is made from components with certain required features and properties. To specify the geometry of the path, you use a 3D sketch.

This chapter describes:

- Connection points and Route points
- Creating the necessary parts
- Routing options
- Creating and editing a routing sub-assembly

The piping functionality is supplied as an add-in application. To enable the functionality, click **Tools, Add-Ins**, and select the **Piping** check box.

## Overview

---

A routing sub-assembly is always a top-level assembly component. Whenever you insert a component with routing information into an assembly, a routing sub-assembly is created for you. Unlike other types of sub-assemblies, you do *not* create a routing assembly in its own window, and then insert it as a component in the higher-level assembly.

The SolidWorks software supports routing functionality for fabricated pipes. You model the path of the pipe by creating a 3D sketch of the pipe centerline. The software uses the centerline definition to generate the pipe and elbow components for the route.

The software makes extensive use of design tables to create and modify the configurations of routing components. The configurations are distinguished by different dimensions and properties. If you are unfamiliar with these concepts, see **Design Tables** on page 8-8.

A pipe part contains a configuration for each type and size of raw stock. As you create and edit the route, a new configuration is generated automatically for each unique cut length of the selected stock. The configurations are saved in a new pipe part; the original pipe part in the library folder is not changed.

Where there are bends in the path, elbows are added automatically. You specify a default elbow fitting to be used at each bend in the route. If you want, you can also let the software create custom elbows at bends whose angle is different from the standard bend angle.

You can add various types of fittings to the route, such as flanges, tees, crosses, and reducers. The fitting components must have configurations that match the pipe sizes.

## General Procedure

The following describes a general procedure for developing the route for a pipe. A more detailed description can be found on page 13-14.

- 1 Create the part documents for the pipe and the fittings (elbows, flanges, and so on).
- 2 Create an assembly with the components to be connected (tanks, pumps, and so on). Add any other components that are needed to specify the path of the route. This includes attachment components such as supporting brackets, obstacles that the routing path must avoid, and so on.
- 3 Position the fixed components using mates, dimensions, or relations to a layout sketch, and save the assembly.
- 4 Insert the necessary end fittings, and mate them to the components you need to connect. A new routing sub-assembly containing the end fittings is created.
- 5 Name the new routing sub-assembly, and specify the route properties (pipe part, elbow part, and other options as described later in this chapter).
- 6 Create a 3D sketch to specify the path. The 3D sketch should include sketch fillets at sharp corners (virtual sharps of two straight segments) and any other bend locations that require elbows.

7 Add tees, crosses, and so on, as needed.

When you exit the 3D sketch, the software generates the pipe segments, inserts the elbows, and connects all the components into a continuous pipe assembly.

Each component in the routing assembly is parametrically related to the 3D sketch. If you change the sketch in any way, the pipes and fittings are updated automatically.

You do not need mates or other relations between the components of the routing sub-assembly, because the sizes and positions are driven by the 3D sketch.

## 3D Sketch

You use a 3D sketch to define the centerline of the route path.

When you drag a flange into an assembly, a new routing sub-assembly is created, and the 3D sketch is started automatically. You can also begin the 3D sketch manually, if the fittings are already in the assembly. Right-click the connection point in a fitting and select **Start Route**.

Sketch the lines, and add fillets as needed. If you select the option **Automatically create sketch fillets**, a fillet is added at each virtual sharp intersection, using the default radius specified in the **Route Properties**.

For information about creating a 3D sketch, see **3D Sketching** on page 2-50.

## Required Features in Piping Parts

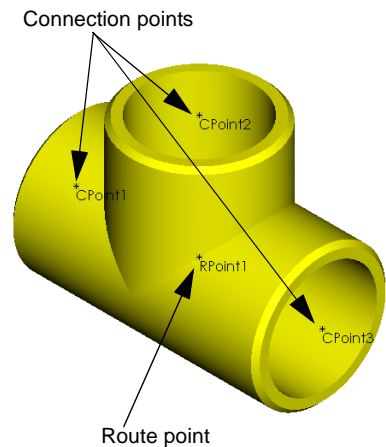
---

There are two special features of the components used in routing sub-assemblies: connection points and route points.

- ❑ A *connection point* is a point in a fitting where a route begins or ends. A pipe segment can only be generated when at least one end is attached to a connection point. Each fitting part must contain a connection point for each port, positioned where you want the adjoining pipe to begin or end.
- ❑ A *route point* is a point in a fitting that is used to position the fitting at an intersection or endpoint in the 3D sketch.

Some types of fittings use both connection points and route points. For example, the tee shown here has three connection points (one for each port), and one route point (at the intersection of the straight run and the outlet).

The specific requirements for each type of fitting are discussed in **Routing Components** on page 13-7.



## Connection Points

### To create a Connection Point:

- 1 Create a sketch point to use for locating the connection point.



---

**NOTE:** The location of the connection point defines the *end* of the adjoining pipe.

- If the pipe extends into the fitting, the connection point must be at the correct depth *inside* the fitting.
- If the pipe is welded to the end face of the fitting, the connection point must be coincident with the end face of the fitting.

The distance between the connection points in the route determines the cut lengths of the pipe segments.

---

- If there is already a sketch of a circle in the correct location, you can use the centerpoint of that circle. For example, if a circular feature is extruded from the face where the pipe ends, you can use the centerpoint of the sketched circle.
  - Otherwise, open a sketch, and add a **Point** . Add dimensions and relations as needed to ensure that the point is concentric with the cylindrical face of the part and positioned at the correct depth.
  - One sketch can contain the sketch points for multiple connection points. In the tee shown earlier in this section, the sketch points for all three connection points and the route point are in a single sketch; the plane of the sketch is coincident with the temporary axes of the three ports.
- 2 Click **Connection Point**  on the Routing toolbar. (If the Routing toolbar is not displayed, click **View, Toolbars, Routing**.)
  - 3 Click the planar face where the pipe exits the fitting, and click the point from Step 1. An arrow is displayed, pointing in the direction that the pipe exits the fitting. If the arrow is pointing in the wrong direction, click the **Reverse direction** check box.
  - 4 Under **Configurable properties**, enter the **Nominal diameter** for the route.
  - 5 Optionally, you can define a **Specification** for each connection point. See the next section for more information about this parameter.
  - 6 Click **OK**.

The **Connection Point** feature contains an **Axis**. When the sketch point was projected onto the face (in Step 3), an axis was created normal to the face at that point. When the fitting is used in a routing sub-assembly, the pipe segment is aligned to this axis.

## Connection Point Parameters in the Design Table

You use a design table to control the dimensions in the configurations of the fittings. In addition, you can use the design table to control the parameters of the connection points for each configuration.

See **Design Tables** on page 8-8 for more information about using design tables to create configurations.

### Diameter

When you create a new connection point, you must specify the **Nominal diameter** of the route. When you create additional configurations of a fitting containing a connection point, each configuration needs an appropriate value for the **Nominal diameter**.

---

**NOTE:** The **Diameter** parameter cannot be driven by equations. You must specify an explicit value for each connection point in each configuration of the fitting.

If you do not add this parameter to the design table, you must edit the definition of each connection point in each configuration in the part document, and specify the appropriate **Nominal diameter**.

---

To add this parameter to the design table, use this syntax (in the header cell):

**Diameter@CPoint*n***

where **CPoint*n*** is the name of the connection point. If the part has multiple connection points, create a column in the design table for each connection point.

In the body cell for each configuration, enter the value (the nominal diameter). In a normal elbow part (not a reducing elbow), the value is the same for both connection points. In a reducer part, the values are different for the two connection points.

To verify that the correct value is used in each case, double-click the **CPoint** feature in the FeatureManager design tree. The **Diameter** is displayed in the graphics area.

The **Diameter** parameter in the connection points of fittings corresponds to the **NominalDiameter@FilterSketch** dimension in the pipe part (see **Pipe Parts** on page 13-7).

### Specification

When you create a new connection point, you can define an optional parameter that identifies the wall thickness of the fitting (the **Specification**). You can either use the default name for this parameter or enter a different **Design table name**. For example, if you use ANSI standards, you could enter **Schedule** as the **Design table name**.

You also specify a **Value**, which can be a dimension or other text, according to the standards you use to identify pipe and fitting thicknesses. You can use the same value in more than one configuration.

When you create additional configurations, each configuration needs to specify the appropriate value for the **Specification** parameter.

If you use the default name for **Specification**, the parameter uses this syntax:

**Specification@CPointn**

– or –

**Schedule@CPointn** (if you use **Schedule** as the **Design table name**)

where **CPointn** is the name of the connection point.

In the body cell for each configuration, enter the value of the specification. Consult a mechanical design handbook for tables of standard dimensions and specifications, or use the standards established within your company.

If the connection points in your routing parts include the optional parameter, you can select the **Use specification** check box in the **Fabricated Piping** dialog box. See Step 5 on page 13-14 for more information.


The **Specification** parameter in the connection points of fittings corresponds to the **\$PRP@Pipe Identifier** property in the pipe part (see **Pipe Parts** on page 13-7).

## Route Points

A **Route Point** is used to position a fitting at an intersection or end point in the 3D sketch.

In a fitting with multiple ports (such as a tee or cross), create a sketch point at the intersection of the axes of the fitting before adding a **Route Point**.

### *To create a Route Point:*

- 1 Click **Route Point**  on the Routing toolbar. (If the Routing toolbar is not displayed, click **View, Toolbars, Routing**.)
- 2 Click a point on the model.
  - In a fitting with multiple ports, click the sketch point at the intersection of the axes.
  - In a flange, click a point that is concentric with the cylindrical faces of the part. If the flange mates with another flange, select a point on the mating face.



# Routing Components

## Pipe Parts

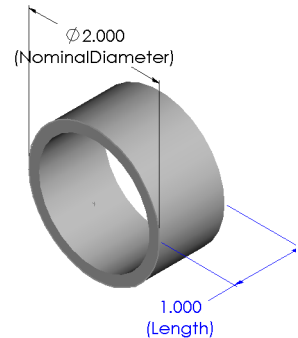
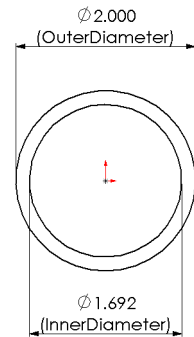
In a pipe part, each type and size of raw stock is represented by a configuration. In the routing sub-assembly, the individual segments are configurations of the pipe part, based on the nominal diameter, the pipe identifier, and the cut length.

**IMPORTANT:** Be sure to use the exact names specified in the following procedure for sketches, dimensions, and properties of the pipe part.

### To create a new pipe part:

- 1 Open a new part, and create a sketch. Name the sketch **PipeSketch**.
- 2 Sketch and dimension two concentric circles. Name the dimensions **OuterDiameter** and **InnerDiameter**.
- 3 Extrude the sketch to any length, then change the name of the **Base-Extrude** feature to **Extrusion**.
- 4 Name the dimension of the extrusion depth **Length**.
- 5 Select one of the planar faces of the pipe, and open a new sketch. Name the sketch **FilterSketch**.
- 6 Sketch another circle, and add a concentric relation to one of the circular model edges. Dimension the circle, and name the dimension **NominalDiameter**. Set the diameter of the circle to the nominal diameter of the pipe.  
This dimension corresponds to the connection point parameter called **Diameter** in the fitting parts.
- 7 Close the sketch.
- 8 Insert a design table to create the configurations. In the header row, include the parameters shown in this example.

PipeSketch sketch



Consult a mechanical design handbook or your company standards for tables of standard dimensions.

	A	B	C	D	E
2		OuterDiameter @PipeSketch	InnerDiameter @PipeSketch	NominalDiameter @FilterSketch	\$PRP@ Pipe Identifier
3	1 inch schedule 40	1.315	1.049	1	schedule40
4	1 inch schedule 80	1.315	0.957	1	schedule80
5	2 inch schedule 40	2.375	2.067	2	schedule40
6	2 inch schedule 80	2.375	1.939	2	schedule80

The parameter **\$PRP@Pipe Identifier** is a property that identifies the wall thickness (and other properties, such as material, if desired) of the pipe. This property corresponds to the **Specification** parameter of connection points in fitting parts.

You can include additional parameters as needed, for properties such as weight per unit of length, cost, part number, and so on. For more information about creating and using design tables, see **Design Tables** on page 8-8.

- 9 Save the part in the folder specified as the **Library folder path** under **Tools, Options, Routing**.

## Elbow Parts

The software inserts an elbow automatically wherever there is a fillet in the 3D sketch. You can also add elbows manually. See **Adding Fittings Manually** on page 13-17.

An elbow part can contain configurations for many different types and sizes of elbow. If you want, the software can create custom elbow parts where needed. See **Routing Options** on page 13-12 for information about this option.

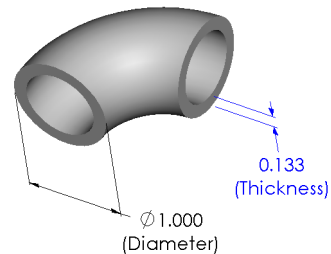
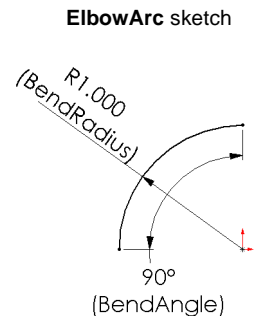
**IMPORTANT:** Be sure to use the exact names specified in the following procedure for sketches, dimensions, and properties of the elbow part.

### To create a simple elbow part:

- 1 Open a new part and create a sketch. Name the sketch **ElbowArc**.
- 2 Sketch and dimension an arc with the radius and angle of a standard elbow. Name the dimensions **BendRadius** and **BendAngle**.

These dimensions correspond to the fillet radius and the angle between the segments in the 3D sketch.

- 3 Add relations as needed to fully define the sketch. For example, in this sketch, there is a coincident relation between the arc centerpoint and the origin, and a horizontal relation between the endpoint and the origin.
- 4 Open a new sketch on a plane that is normal to one end of the arc. Name the sketch **Route**.
- 5 Sketch and dimension a circle. Name the diameter dimension **Diameter**. Set the diameter of the circle to the *outside* diameter value of the elbow. Add a concentric relation between the endpoint of the arc and the centerpoint of the circle.
- 6 Insert a **Sweep** feature using the two sketches.
- 7 Insert a **Shell** feature that removes both end faces of the elbow, with an appropriate **Thickness** for the walls of the part.



- 8 Create **Connection Points** where the pipe exits the elbow at both ends. See **Connection Points** on page 13-4.
- 9 Insert a design table to create the configurations. In the header row, include the parameters shown in this example:

	A	B	C	D	E	F-I
2		Diameter @Route	BendRadius @ElbowArc	BendAngle @ElbowArc	Thickness @ Shell1	
3	1 inch schedule 40	1.315	1.000	90	0.133	...
4	1 inch schedule 80	1.315	1.500	90	0.179	...
5	2 inch schedule 40	2.375	2.000	90	0.154	...
6	2 inch schedule 80	2.375	2.500	90	0.218	...

	A	B-E	F	G	H	I
2			Diameter @CPoint1	Specification @CPoint1	Diameter @CPoint2	Specification @CPoint2
3	1 inch schedule 40	...	1.000	schedule40	1.000	schedule40
4	1 inch schedule 80	...	1.000	schedule80	1.000	schedule80
5	2 inch schedule 40	...	2.000	schedule40	2.000	schedule40
6	2 inch schedule 80	...	2.000	schedule80	2.000	schedule80

The columns for **Specification@CPointn** are optional. The columns for **Diameter@CPointn** are required. See **Connection Point Parameters in the Design Table** on page 13-5.

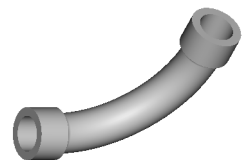
Notice that the values are the same for both **CPoint1** and **CPoint2**.

You can include additional parameters as needed, for properties such as part number, cost, weight, and so on. For more information about creating and using design tables, see **Design Tables** on page 8-8.

- 10 Save the part in the folder specified as the **Library folder path** under **Tools, Options, Routing**.

You can create other types of elbows with additional features.

Create a simple elbow with a **Sweep** feature as described in Steps 1-7. Add features as needed before or after the **Shell** feature. Then add the connection points and the design table.

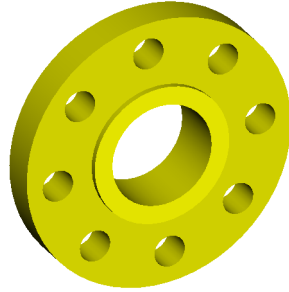


## Flange Parts

A flange is often used at the ends of a piping run to connect the pipe to a fixed component, such as a pump or tank. Flanges can also be used to connect long straight sections of pipe.

A flange part must have one **Connection Point** and one **Route Point**. The **Route Point** must be collinear with the axis of the **Connection Point**, and therefore, concentric with a circular edge of the fitting.


The **Route Point** is used to position the flange; the pipe ends at the **Connection Point**.



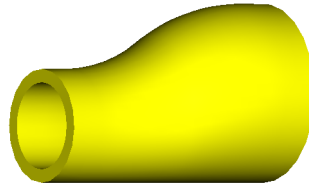
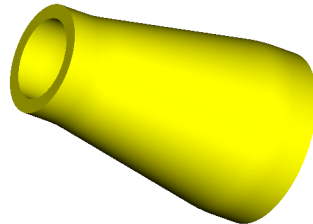
## Reducers

A reducer is used to change the diameter of the pipe at a selected location.

A reducer has two **Connection Points** with different values for the **Diameter** parameter.

A reducer also has one **Route Point** (usually in the middle of the part) that positions the fitting at a selected point in the 3D sketch. The point in the 3D sketch can be an intersection of collinear segments, or a point that you insert using the **Split Curve** tool  on the Sketch Tools toolbar.

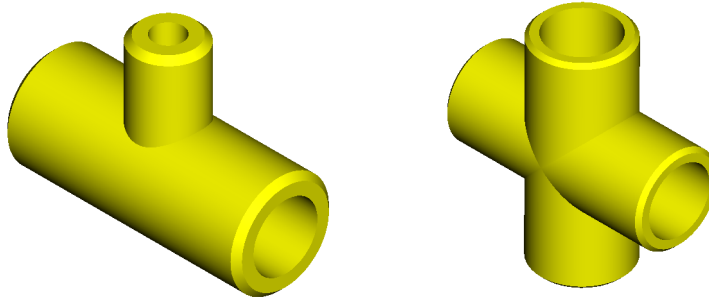
The route point in an offset (eccentric) reducer must be axially aligned with one of the connection points. That axis is used to align the reducer with the 3D sketch segment. You may need to create two versions of these reducers: one whose route point is aligned with the larger port, and one whose route point is aligned with the smaller port.



When you insert the reducer at the end of a sketch segment, select the version that aligns the fitting as needed. To continue the route from the other end of the reducer, right-click the **Connection Point**, and select **Start Route**.

## Other Fittings

You can add tees, Ys, crosses, and other multi-port fittings at intersections in the 3D sketch. Fittings with multiple branches must have a **Connection Point** at each port, and a **Route Point** at the intersection of the branches.



When you insert the fitting, the **Route Point** is placed coincident with the intersection in the 3D sketch.

The connection points can have different **Diameters** and **Specifications** if needed.

---

**NOTE:** The straight run for a tee must consist of *two separate segments*, not one continuous segment (because two segments of pipe must be created for the straight run).



---

The straight runs for crosses must also be separate segments.

The number of sketch lines at the intersection can be less than the number of ports in the fitting you want to insert. Insert and align the fitting as desired, then add the remaining sketch segments later. See **Adding Fittings Manually** on page 13-17.

## Routing Options

---

### To specify Routing Options:

- 1 Click **Tools, Options, Routing**.
- 2 Specify the **Library folder path** where the routing components are stored. It is recommended that you store all the components that you use in routing assemblies in the same folder. When you create or edit a routing assembly, the software searches in this folder for the components to use.

You can create additional folders under the **Library folder path** if you want to organize your parts in a specific way (by material, by type, and so on).

Some sample routing parts are supplied with the software, in the following folder (unless you selected a different path for the **data** folder when you installed the SolidWorks software):

***install\_directory\data\Palette Parts\Routing***

When you specify a **Library folder path**, that path is automatically included in the **Folders** list for **Palette parts** (under **Tools, Options, External References**). This allows you to add fittings by dragging them from the Feature Palette window.

See **External References Options** on page A-11 for more information about the **Folders** list.

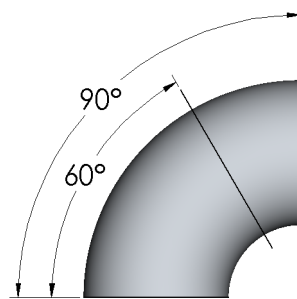
- 3 Select from these options:
  - **Create custom fittings.** When selected, the software automatically creates custom configurations of the default elbow fitting when needed. This is done only in cases where a standard elbow configuration can be cut down to create a custom elbow.

To create a custom configuration, the **BendAngle** (the angle *through which* the material is bent) in the custom elbow must be *less than* the **BendAngle** in the standard elbow.

For example, an elbow with a **BendAngle** of 90° can be cut down to create a 60° elbow.

The software *does not* create custom elbows for variations in the **BendRadius** or **Diameter**.

If your company standards do not permit cut-down elbows, clear the **Create custom fittings** check box. You may need to modify the 3D sketch to use only fillets with standard bend angles.

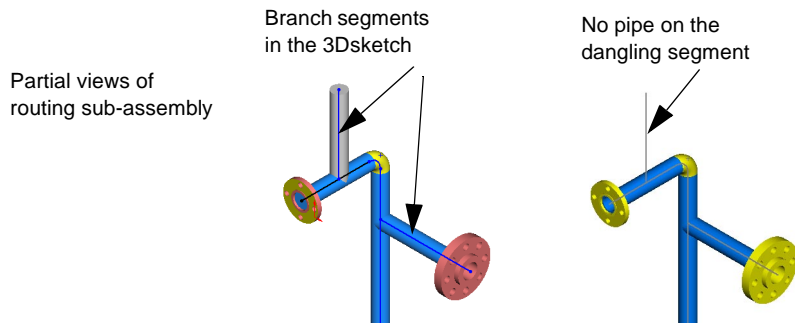


- **Automatically create sketch fillets.** Select this check box if you want the software to automatically add fillets at intersections as you sketch. The fillet radius is based on the **BendRadius@ElbowArc** dimension in the elbow part that you select for the route. This option applies only to 3D sketches that are used as paths of routing assemblies.

- **Create pipes on open line segments.** Select this check box if you want a pipe to be generated for 3D sketch segments that are connected to a fitting at *only one* end. For example, if you have not yet added a flange at the end of the route, the last segment of the sketch (beyond the last fitting in the run) is an open segment. If this check box is cleared, no pipe is generated on the open segment.

Pipes are never generated for segments that are not connected to a fitting at either end. For example, when you create a t-junction in the sketch, the branch is connected to the straight run segment at one end, but not to a fitting. You must add a fitting at the other end of the branch, or add a fitting at the intersection, to create a pipe for the branch.

In this example, a pipe is generated for the branch with a flange fitting, but no pipe is generated for the branch without a fitting.



4 Click **OK**.

## Creating a Route

---

There are two ways to start a route.

- ❑ You can begin a route immediately upon inserting a flange or other fitting into a saved assembly document. The newly inserted flange becomes the first component of a new routing sub-assembly
- ❑ You can insert a flange, but not start the route immediately. Mate the flange, then start the route from a connection point on the flange when you are ready. In this case, the flange is not a component of the routing sub-assembly.

### **To create a route:**

- 1 Click **Tools, Options, Routing**, and make sure that the options are set correctly for the route you want to create. See **Routing Options** on page 13-12.
- 2 Create and save a new assembly document.
- 3 Insert the components that you want to connect, and add mates to position the components as needed. Include any components that the pipe needs to go around as well.
- 4 Drag a flange into the assembly from the Feature Palette window, from an open part window, or from Windows Explorer. Select the configuration to use, and click **OK**.

The **Fabricated Piping** dialog box is displayed.

- 5 In the **Fabricated Piping** dialog box, specify the properties of the route:
  - Under **Start or edit**, specify the **Assembly file name** for the routing sub-assembly document.
  - Under **Connection point properties**, the **Name**, **Nominal diameter**, and **Specification** (if this optional parameter exists in the selected point) of the selected connection point are displayed.

Select the **Use specification** check box if you want software to search in the pipe and fitting documents for configurations with the same **Specification**.

- Under **Pipe fitting**, click **Browse**, and locate the **File name** for the pipe part to use. By default, the software looks in the folder specified as the **Library folder path** (under **Tools, Options, Routing**).

If you selected **Use specification**, and the pipe part contains a configuration whose nominal diameter and specification match the selected connection point, the appropriate **Base configuration** is listed.

Otherwise, the **Base configuration** list contains the names of any configurations that match the nominal diameter of the connection point. Clear the **Use specification** check box, and select a configuration from the list. The pipe part must contain at least one configuration with the appropriate nominal diameter.

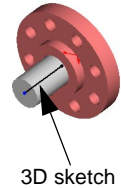
The **Wall thickness** of the pipe is displayed for reference only.




- Select an **Elbow fitting** part in the same way as you selected the pipe part. The default **Bend radius** of the elbow is displayed for reference only.
- To save the current specifications and use them as the defaults in any new assemblies that you create, click the **Save settings** check box.
- Click **Options** if you want to override the default settings (from **Tools, Options, Routing**) for the current route. See **Routing Options** on page 13-12.

- 6 Click **OK**. Switch to **Isometric** view if necessary.

A new routing sub-assembly is created as a top-level assembly component. A 3D sketch for the route is started



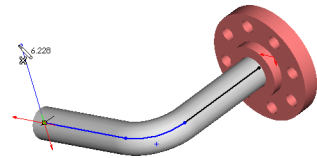
- 7 If the flange is mated correctly by smart mates, skip to Step 11.


If the flange is not mated the way you want it to be (if you did not add smart mates while inserting the flange), exit the 3D sketch now. Right-click anywhere in the graphics area and select **Exit Sketch**, or click the **3D Sketch** tool  on the Sketch toolbar.

- 8 Save the pipe part with a new name. The configurations that are created for each cut length in the route are stored in this file.

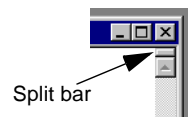
- 9 Mate the flange as needed.

- 10 Open the 3D sketch for editing. Right-click the **Route** feature in the route sub-assembly, and select **Edit Route**.




- 11 Sketch the path of the route using the **Line** tool  on the Sketch Tools toolbar. Press **Tab** to change from one sketch plane to another.

To help you visualize the sketch in 3D space, you may find it useful to split the graphics window. Try displaying an isometric view in shaded mode in one pane, and a front or top view in wireframe mode in the other pane. To split the window, drag the split bars at the corners of the window.



Add dimensions and relations as needed. For example, to create a t-junction between segments on two different sketch planes, add a coincident relation between the straight run segment and the endpoint of the branch segment. For more information, see **3D Sketching** on page 2-50.

If you selected the **Automatically create sketch fillets** option, fillets are added automatically at intersections as you sketch. If you want to add sketch fillets manually (for example, when you need a bend radius that is different from the default), use the **Fillet** tool  on the Sketch Tools toolbar.

To display the virtual sharp intersections in the sketch, click **Tools, Options, General**, and select the **Display virtual sharps** check box.

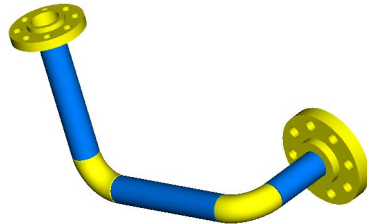
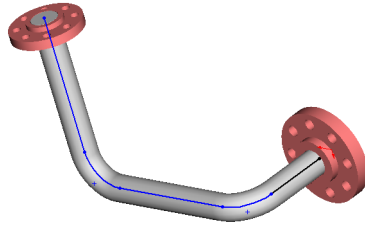
- 12 Add fittings as needed. See **Adding Fittings Manually** on page 13-17.
- 13 Exit the sketch. (See Step 7.)
- 14 If you did not already name the pipe file in Step 8, name it now. Each unique cut length is a configuration of the new pipe part.

If any custom elbow fittings are required, each fitting is saved as a separate new part. You are asked to specify a name for each custom fitting.

For more information about creating custom elbow fittings, see **Routing Options** on page 13-12.

The pipe segments and the elbows are added to the routing sub-assembly.

- 15 To return to editing the top-level assembly, right-click and select **Edit Assembly**.



## Editing a Route

---

### Editing the 3D Sketch

*To edit the 3D sketch and modify the route:*

Right-click the **Route** feature and select **Edit Route**.

You can also right-click the **3DSketch** feature in the route sub-assembly in the FeatureManager design tree, and select **Edit Sketch**.


As you drag segments in the sketch, the pipe updates to reflect the changes. When you exit the 3D sketch, the sub-assembly rebuilds.

## Adding Fittings Manually

Elbows are added automatically at fillets in the 3D sketch. You can add other types of fittings, such as flanges, tees, crosses and so on, manually.

### *To add a fitting manually:*

- 1 While editing the 3D sketch, drag a fitting from the Feature Palette window and point to the location in the 3D sketch where you want to place the fitting.

The location can be an endpoint or an intersection, or a point that you inserted in the middle of a segment, using the **Split Curve** tool  on the Sketch Tools toolbar.

- 2 Press **Tab** to cycle through the possible orientations of the fitting. When the appropriate alignment is displayed, drop the fitting.
- 3 Select the desired configuration, and click **OK**.

– or –

- 1 While editing the 3D sketch, right-click a point in the 3D sketch, and select **Add Fitting** from the shortcut menu.
- 2 Select the fitting from the **Select a Fitting** dialog box. Select the appropriate configuration, and click **OK**.

The fitting is positioned with its **Route Point** at the selected vertex. The software attempts to align the fitting component correctly with the sketch lines.

### *To change the alignment of a fitting:*

If a fitting is not aligned correctly with the sketch, you can change the alignment.

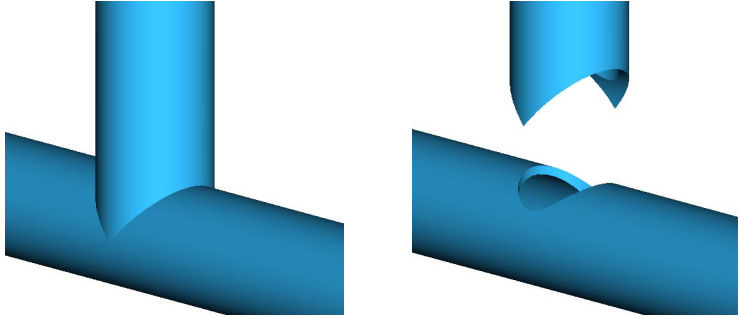
- 1 Exit the 3D sketch.
- 2 Right-click the fitting and select **Change fitting alignment**. Notice the pointer.
- 3 Click the right-mouse button or press **Tab** to cycle through the possible alignments.
- 4 When the desired alignment is displayed, click the left-mouse button or press **Enter**.



## Pipe to Pipe Penetration

If cost or space considerations demand, fittings such as tees can be eliminated by welding the pipes at the intersection.


In the exploded view shown here, you can see that the branch (the pipe that intersects the straight run) is cut using the profile of the straight run. The straight run is cut using the inner profile of the branch.

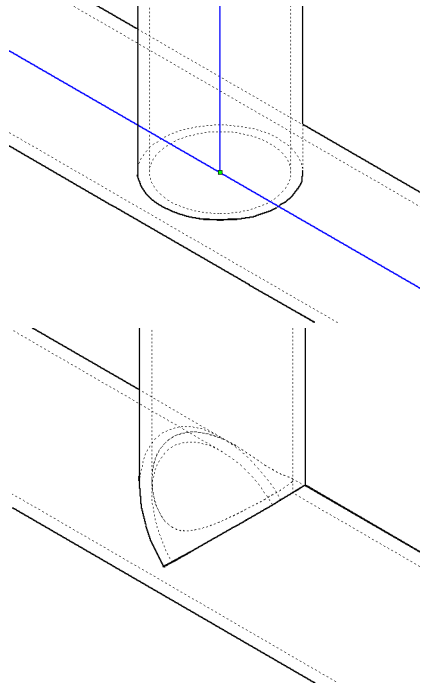


In general, you should perform all editing of the route in the context of the top-level assembly that contains the routing sub-assembly. However, to perform pipe to pipe penetration, you must edit the routing sub-assembly in its own window.

### ***To cut pipes that penetrate each other:***

- 1 Right-click the routing sub-assembly in the FeatureManager design tree, and select **Open route\_assembly.sldasm**.
- 2 Right-click the **Route** feature, and select **Edit route**.
- 3 Right-click the point where the pipes intersect, (the intersection of the 3D sketch segments), and select **Penetrate**.

The cuts are created as in-context features of the routing sub-assembly. Notice the **Update Sketch**  features at the bottom of the Feature Manager design tree).



---

**NOTE:** For pipe to pipe penetration, the straight run must be a *single* segment in the 3D sketch, because the straight run is a single segment of pipe. The endpoint of the branch segment must be coincident with the segment for the straight run. You cannot penetrate at an intersection of three segments.



---

## Reorganizing a Routing Sub-assembly

---

You can use the assembly restructuring functionality to break down large pipe assemblies into manageable proportions.

- ❑ You can *form a new routing sub-assembly* from selected components in a routing sub-assembly. With the top-level assembly active, select one or more components, then right-click the desired components and select **Form New Sub-assembly Here**. The new sub-assembly is a routing sub-assembly, but it does not contain a route of its own. The new sub-assembly is driven by the route of its parent assembly.

You can create multiple nested sub-assemblies in a routing sub-assembly, with as many levels as you want.

- ❑ You can *move a component of a routing sub-assembly* to a parent or child assembly that uses the same 3D sketch to drive the route. You can either drag and drop the components, or use **Tools, Reorganize Components**. You cannot move a routing component outside of its own branch of the hierarchy. You cannot move components from outside the routing sub-assembly into the routing sub-assembly.
- ❑ You can *dissolve a routing sub-assembly* if its parent is a routing sub-assembly. Right-click the nested sub-assembly, and select **Dissolve Sub-assembly**. You cannot dissolve a routing sub-assembly that contains a route and its 3D sketch; therefore, you cannot dissolve the routing sub-assembly that is the top-level assembly component.

See **Working with Sub-assemblies** on page 6-28 for more information about reorganizing assembly components.



## Importing and Exporting Files

---

This chapter describes these SolidWorks file import and export topics:

- ❑ Importing and exporting the following file types:
  - IGES
  - Parasolid text and binary
  - STEP
  - ACIS™
  - VDAFS
  - STL (Export only)
  - DXF/DWG
  - VRML
  - TIFF
- ❑ Setting import and export options
- ❑ Recognizing features with FeatureWorks™

## Importing/Exporting SolidWorks Documents

---

You can import files to the SolidWorks software from other applications. You can export SolidWorks documents to a number of formats for use with other applications. The following table displays the data translation methods available for SolidWorks documents:

**Table 14-1. Import and Export File Types**

	Parts		Assemblies		Drawings	
	Import	Export	Import	Export	Import	Export
IGES	√	√	√	√		
Parasolid	√	√	√	√		
STEP	√	√	√	√		
ACIS	√	√	√	√		
VDAFS	√	√				
STL		√		√		
DXF/DWG	√				√	√
VRML	√	√	√	√		
TIFF	√	√	√	√		√

### Importing Documents

Before you import a document to the SolidWorks software, make sure that you have the import options set for your purposes. Refer to **Setting Import Options** in the next section.

***To import a file from another application:***

- 1 Click **File, Open**.  
The **Open** dialog box appears.
- 2 In the **Files of type** list, click the desired format.
- 3 Browse to the desired file, and click **Open**.  
The selected file is opened.

***To import a surface from another application:***

You can import surface geometry from an IGES, ACIS, STEP, VDAFS, or VRML file as reference geometry into an existing part document.

- 1 In a part document, click **Insert, Surface, Imported**.  
The **Open** dialog box appears.
- 2 Browse to the desired file, and click **Open**.



One or more surfaces are imported, and a **Surface** feature is added to the part. The surface is positioned relative to the origin, using the coordinates in the imported file. For information about using imported surfaces to create solid model features, see **Using Surfaces to Create Features** on page 4-56.

---

**NOTE:** Wireframe geometry and curves are ignored when importing surfaces as reference geometry.

---

#### **Notes on importing documents:**

- TIFF files can be imported as the background image of a SolidWorks part or assembly document using **Insert, Picture**. See **To import a TIFF file:** on page 14-18.
- In an IGES file, if there are curves or if there are different levels (or layers), the **IGES-In Surfaces, Curves, and Levels** dialog box appears. See **IGES Surfaces, Curves, and Levels** on page 14-9.
- If importing a DXF/DWG file, the **Open DXF/DWG File** dialog box appears after you click **Open**. See **Open DXF/DWG File** on page 14-13.
- When importing an assembly document with 100 or more components, the components are suppressed as they are opened to improve import performance. This works for IGES, Parasolid, STEP, and ACIS.
- An error file (**.err**) that lists any import errors is written to the same directory as the imported file. This is true for IGES, STEP, ACIS, and VDAFS. For IGES only, if there are unsupported entities present in the IGES file, a report file (**.rpt**) is written as well.

## **Setting Import Options**

Before you import an IGES, STEP, ACIS or VDAFS file, make sure the options are set the way you want. The import options apply only to these surface-based translators.

#### **To set import options:**

- 1** Click **File, Open**.
- 2** In the **Files of type** list, click the desired format.
- 3** Click **Options**.  
The **Import Options** dialog box appears.
- 4** **Error resolution.** Select the default method that you want the software to use to resolve problems.
  - The solutions for **Surface errors**, **Surface flaws**, and **Significant digits** (IGES import only) are:
    - Prompt** (ask for a solution each time a problem occurs)
    - Try to knit into solid**
    - Try to form reference surfaces**
    - Cancel operation**

- The solutions for **Blanked surfaces** (IGES import only) are:

**Prompt**

**Ignore blanked surfaces**

**Do NOT ignore blanked surfaces**

**Cancel operation**

---

**NOTE:** **Surface errors** occur when a surface cannot be imported.

**Surface flaws** occur when a surface can be imported but has some sort of defect.

**Significant digits** problems occur when the data contains low precision data.

**Blanked surfaces** are invisible surfaces included in a file, typically, for reference purposes.

---

- 5 **Try forming solid model(s) from surfaces.** When the check box is cleared, surfaces are automatically grouped into one or more surface features (named **Surface 1, 2, ...**).
- 6 Click **OK**.

## Editing Imported Features

You can replace an imported feature with geometry from a new file.

### *To edit a feature created from an imported document:*

- 1 In the FeatureManager design tree, right-click the feature created from the imported document, and select **Edit Definition**.

The **Open** dialog box appears.

- 2 In the **Files of type** list, click the desired format.
- 3 Browse to the desired file, and click **Open**.

An imported solid is only replaced if the data in the new document can be successfully knitted into a body. A surface feature is replaced with the first surface in the new document, and subsequent surfaces in the new file are added to the model.

---

**NOTE:** You can only edit features that were created from an IGES, STEP, ACIS, VRML or VDAFS file.

---

## Import Diagnosis

You can diagnose and repair any gaps or bad faces on imported features. Candidates for import diagnosis are:

- an imported solid body that has rebuild errors.
- an imported surface that was not able to knit into a solid body.

### *To run an import diagnosis and to repair an imported feature:*

- 1 Right-click the **Imported** feature in the FeatureManager design tree, and select **Import Diagnosis**.

The **Import Diagnosis** dialog box appears.

- 2 Under **Gaps**, any existing gaps and gap edges are listed.
  - Click **Gap** or **Gap edge**, then click **Previous** or **Next** to view the gaps or gap edges.
  - Click **Close All Gaps** to repair the gaps.

If all of the gaps are closed, “**No gap found,**” appears in the dialog box.

- 3 Under **Faces**, any existing bad faces are listed.
  - Select a face in the list to highlight the face in the graphics area.
  - Click **Remove Faces** to remove the bad faces and to create gaps in the feature.
  - Click **Fix Faces** to fix the bad faces.

If all of the bad faces are repaired, “**No bad faces found,**” appears in the dialog box.

- 4 Click **OK**.

## Exporting Documents

**To export a SolidWorks document with a new format:**

Locate the column for the export format you want. Go down the column and complete each instruction that is marked with a check mark ✓.

	IGES	STEP	ACIS	Parasolid	VRML	TIFF	VDAFS	STL	DXF/DWG
Select the faces or surfaces of a part, or select the components of an assembly, for export. If nothing is selected, the entire part or assembly is exported.	✓	✓	✓	✓			✓		
Click <b>File, Save As</b> .	✓	✓	✓	✓	✓	✓	✓	✓	✓
In the <b>Save as type</b> list, click the desired format.	✓	✓	✓	✓	✓	✓	✓	✓	✓
Make sure that the options are appropriate for the target application. Select from the options on the <b>Save As</b> dialog box, or click <b>Options</b> . For details about these options, refer to the sections later in this chapter.	✓		✓	✓	✓	✓		✓	✓
In the <b>File name</b> box, type a name. The extension for the selected format is automatically added.	✓	✓	✓	✓	✓	✓	✓	✓	✓
In the <b>Output coordinate system</b> list, select a coordinate system to apply for export. If you select <b>-- default --</b> , no transformation matrix is applied.	✓	✓	✓	✓			✓	✓	
Click <b>Save</b> .	✓	✓	✓	✓	✓	✓	✓	✓	✓
If you selected any faces or surfaces of a part, indicate whether you want to export the selected items or the entire model, then click <b>OK</b> .	✓	✓	✓	✓			✓		
Before you can export an assembly, you must fully resolve any lightweight parts that it contains. See <b>Lightweight Parts</b> on page 6-40 and <b>Loading Additional Model Data</b> on page 6-41 for more information.	✓	✓	✓	✓	✓	✓		✓	

## IGES Files (.igs, .iges)

---

The following IGES entity types are supported for import and export, except where noted:

<b>IGES Entity Type</b>	<b>Entity Name</b>
416	External reference entity
408	Singular subfigure instance entity
406 Form 12	External reference file
402 Form 7	Group entity
314	Color definition entity
308	Subfigure definition entity
144	Trimmed (parametric) surface
143 Import only	Bounded surface entity
142	Curve on a parametric surface
140 Import only	Offset surface entity
128	Rational B-spline surface
126	Rational B-spline curve
124	Transformation matrix entity
122	Tabulated cylinder
120	Surface of revolution
118 Import only	Ruled surface
114 Import only	Parametric spline surface
112	Parametric spline curve
110	Line
106 Form 12 Import only	Copious Data - 3D piecewise linear curve
106 Form 11 Import only	Copious Data - 2D piecewise linear curve
104 Import only	Conic arc
102	Composite curve
100	Circular arc

## Exporting IGES Files

You can export SolidWorks part and assembly documents to IGES format. Parts and surfaces exported to IGES format retain their part or surface color, and are displayed in color when in **Shaded** mode.

### IGES Export Options

#### □ Output as

- **Trimmed surfaces.** The faces of the part, assembly, or the selected surfaces, convert to trimmed surfaces (Entity Type 144) in the IGES file.

The IGES entity types that compose the trimmed surfaces depend on the export format chosen. The following table shows the IGES entity types that compose the trimmed surfaces in the **Surface representation** list:

Surface representation	Exported IGES Entity Types
STANDARD	144, 142, 128, 126, 122, 120, 110, 102, 100
NURBS	144, 142, 128, 126, 110, 102, 100
ANSYS	144, 142, 128, 126, 110, 102, 100
COSMOS	144, 142, 128, 126, 110, 102, 100
MASTERCAM	144, 142, 128, 126, 110, 102, 100
SURFCAM	144, 142, 128, 126, 110, 102, 100
SMARTCAM	144, 142, 128, 126, 110, 102, 100
TEKSOFT	144, 142, 128, 126, 110, 102, 100
ALPHACAM	144, 142, 128, 126, 110, 102, 100

---

**NOTE:** If the system you are exporting to is not listed, refer to the documentation for that system to see which entities are supported, and choose an appropriate setting.

---

- **3D curves.** The solid body converts to a 3D wireframe representation in the IGES file. Unlike surfaces or faces, you cannot export individually selected model edges to an IGES file.

Select either **B-Splines (Entity 126)** or **Parametric Splines (Entity 112)** depending on the entity types required by the target system. The following table shows which IGES entities are exported for each wireframe selection.

3D Curves	Exported IGES Entity Types
B-splines (Entity 126)	126, 110, 102*, 100
Parametric splines (Entity 112)	112, 110, 102*, 100

\*Exported only if you select the **Duplicate entities** check box.

- **Duplicate entities.** Select this check box if you want to export composite curves (entity type 102) to any of the available export formats. Because each edge is included once in each of two composite curves forming the boundaries of the faces that meet at each edge, the curves are duplicated. For each pair of duplicate curves, one curve has the opposite sense (direction) from the other.
- **High trim curve accuracy.** Select this check box when high curve accuracy is essential. This setting affects files exported both with **Trimmed surfaces** and with **3D curves**. High trim curve accuracy can sometimes help if the target system has trouble importing the IGES file, or cannot knit the surfaces into a useful solid.
- Save all components of an assembly in a single file.** When selected, all assembly components, sub-assemblies, and sub-assembly components are saved in one file. Otherwise, the assembly components and the sub-assembly components are saved as individual IGES files in the same directory.
- Flatten assembly hierarchy.** When selected, the assembly is flattened to one level of part bodies (no sub-assemblies). Components of sub-assemblies are at the same level as top-level components.

## Importing IGES Files

You can import surfaces from IGES files and use them to:

- Create a base feature from a group of IGES surfaces that form a closed volume
- Trim a part with an imported surface
- Create a base, boss, or cut feature by thickening an imported surface

You can import 3D curves from IGES files and use them to:

- Create planes used in pierce point relations in sketches
- Create paths and guide curves for sweeps and lofts

## IGES Surfaces, Curves, and Levels

If there are curves in the IGES file or if there are different levels (or layers) in the IGES file, the **IGES-In Surfaces, Curves, and Levels** dialog box appears when you open an IGES file. Levels are sometimes used by other systems to separate assembly components or to separate IGES entity types.

### *To select the desired entities and levels:*

- 1 Select the **Include surfaces** or **Include curves** check box, or both, to import these items in the file, if applicable.
- 2 Select the levels to include:
  - **Include All Levels.** All levels are imported to level zero.
  - **Include Levels.** Only the specified levels are imported to level zero.
- 3 Click **OK** to import the IGES file.

## Imported IGES Data

If there is wireframe geometry in the file, the software reads the IGES data and forms curves for the IGES wireframe entities. The 2D geometry on the XY plane is imported into a sketch rather than imported as reference curves.

If there are surfaces in the file, they are read as follows:

- If there are blanked surfaces, they are imported and added to the FeatureManager design tree as surface features. Blanked surface import occurs if you have either **Prompt** (and you select **Yes** to blanked surface import) or **Do NOT ignore blanked surfaces** selected in the **Import Options** dialog box.
- If the attempt to knit the surfaces into a solid succeeds, the solid appears as the base feature (named **Imported1**) in a new part file. You can *add* features (bosses, cuts, etc.) to this base feature, but you cannot *edit* the base feature itself.
- If the software finds that the surfaces in the IGES file represent multiple closed volumes, then one part is made for each closed volume. An assembly file also is made that includes the imported parts positioned relative to the assembly origin, according to how the surfaces are defined in the IGES file.
- If the attempt to knit the surfaces fails, the IGES surfaces are grouped into one or more surface features (named **Surface1, 2, ...**) in a new part file.

If you want the software to try to form a solid from the IGES data, export the solid or surfaces from the source system as trimmed surfaces (Entity type 144) or as untrimmed surfaces (Entity types 128, 122, 120, 118, 114). Also, set the number of significant digits (precision) as high as possible to ensure accurate translation.

## Parasolid Files (.x\_t, .x\_b)

---

- You can export SolidWorks part or assembly documents to Parasolid text files or binary files. Binary files are smaller than text files, but binary files are not supported in some target applications. Select the type supported by the target application.
- You can import Parasolid text files or binary files to SolidWorks documents. SolidWorks 99 supports Parasolid version 11.0 and earlier for import.
- Some other systems use the **.xmt\_txt** extension for Parasolid text files and the **.xmt\_bin** extension for Parasolid binary files. Change the extension to **.x\_t** or **.x\_b** before you import the file.
- Data exported to or imported from Parasolid format retains its color when displayed in **Shaded** mode.
- Component names in assemblies are retained for both import and export.
- Curve or point data is not supported for import or export.



## Parasolid Export Options

- Version.** Select the type supported by the target system.
- Flatten assembly hierarchy.** Select this check box to flatten the assembly to one level of only part bodies. A flattened file contains a top-level assembly and a series of parts that contain imported features.

## STEP Files (.step)

---

- You can export or import SolidWorks part or assembly documents to STEP files.
- STEP Application Protocol AP203 is supported for both import and export.
  - Conformance Classes 1, 2, and 6 of AP203 are fully supported.
  - Conformance Classes 4 and 5 of AP203 are partially supported.
- STEP Application Protocol AP214 is supported for import only.

Conformance Classes 1 and 2 of AP214 are partially supported. Only the body geometry and topology information is imported from an AP214 STEP file. The remaining AP214 information in the STEP file is discarded.

## ACIS Files (.sat)

---

- You can export or import SolidWorks part or assembly documents to ACIS™ files.
- For import, if the units of measure are not explicitly specified in the ACIS file, a dialog box gives you the opportunity to specify the units. Files created with early versions of the ACIS modeler do not contain information about the units of measure.

## ACIS Export Options

- Version.** Select the type supported by the target system.
- Units.** Select the default units of measure to use for ACIS files from the list: **millimeters, centimeters, meters, inches, or feet.**

## VDAFS Files (.vda)

---

VDAFS is a neutral file format for the exchange of surface geometry. You can export or import SolidWorks part documents to VDAFS files.

## STL Files (.stl)

---

You can export SolidWorks part and assembly documents to STL format. STL format is intended for transfer to rapid prototyping machines. SolidWorks cannot import STL files.

### STL Export Options

**Output format**

- Select **Binary** or **ASCII**. Binary files are smaller than ASCII files, but they are not supported in some other systems. Select the type supported by the target system.
- **Do not translate STL output data to positive space**. Select this check box to export parts in their original position in global space, relative to the origin.
- **Save all components of an assembly in a single file**. Select this check box to save the assembly and each of its components into a single **.stl** file.
- **Check for interferences**. Select this check box to perform an interference check on an assembly before saving. Exporting an assembly with coincident or interfering bodies to a single **.stl** file produces a file that may be unsuitable for some rapid prototyping systems. It is highly recommended that you check for interference between the components before saving the assembly document. For more information, see **Detecting Interference Between Components** on page 6-38.

**Quality**

**Coarse** and **Fine** are preset values. Choose **Custom** if you want to set the resolution. If you choose **Custom**:

- Adjust the **Total quality** slider to set the **Deviation**. **Deviation** controls whole-part tessellation. Lower numbers generate files with greater whole-part accuracy.
- Adjust the **Detail quality** slider to set the **Angle Tolerance**. **Angle Tolerance** controls smaller detail tessellation. Lower numbers generate files with greater small-detail accuracy.

As you adjust the two sliders, notice how the corresponding concentric circles adjust accordingly. The circles show approximately how the tessellation varies as a result of the settings. (See the **Preview** option.)

---

**NOTE:** Files generated with higher quality settings are larger in size and slower to generate. Experiment with these **Quality** settings to determine the best settings for your own equipment.

---

**Preview.** Select this check box to display a preview of a faceted version of the model in the graphics area (move the dialog box out of the way to view the faceted model.)

**Show STL info before saving.** Select this check box to display a faceted model preview in the graphics area and display a dialog box with the following information: Triangles (number), binary file size, file format, and the directory path and file name. In the information dialog box, click **Yes** to save the file in the indicated location; click **No** to cancel the operation.

## DXF/DWG Files (.dxf, .dwg)

---

You can import a **.dxf** or **.dwg** file to either a part or drawing document. In a drawing document, you can import the geometry to the drawing sheet or the drawing template. Only drawing documents can be saved as **.dxf** or **.dwg** files. When you export a drawing as a **.dxf** or **.dwg** file, the sheet scale of the drawing is used for the new file.

### Open DXF/DWG File

The **Open DXF/DWG File** dialog box appears each time you open a **.dxf** or **.dwg** file. You have the option of importing to either a drawing or a part. If you import to a drawing, the layer, color, and line style information in the **.dxf** or **.dwg** file is also imported.

- If you choose **Import to Drawing**, adjust the drawing sheet size and units of measure, if necessary.
  - Click the **Move entities onto sheet** check box to move the lower left-hand corner of the imported drawing to the origin of the SolidWorks drawing sheet.
  - Click the **Import to template** check box to import the file as a SolidWorks template.
- If you choose **Import to Part**, adjust the units of measure if necessary.

The entities in the **.dxf** or **.dwg** file form 3D curves in the part document.

### Attach Dimensions

In an imported **.dxf** or **.dwg** drawing document, you can attach the dimensions to the nearest reasonable sketch entities all at once.

#### *To attach dimensions:*

With an imported **.dxf** or **.dwg** drawing document active, click **Tools, Dimensions, Attach Dimensions**.

### Sketch from Drawing

You can transfer entities from an existing drawing to an open sketch. You can use this function to create a sketch from a **.dxf** or **.dwg** file imported into a drawing document.

#### *To insert a sketch:*

- 1 Open the drawing that contains the entities you want to insert.
- 2 Open the part where you want to insert the entities.
- 3 Open a sketch in the part, then click **Insert, Sketch from Drawing**.
- 4 From the **Window** menu, select the drawing to make it the active window.
- 5 Select the entities by dragging a rectangle around them.
- 6 Return to the part window. The entities are inserted into the active sketch in the part, and you can now edit the sketch as needed. See **Editing a Sketch** on page 2-31 for more information.

## DXF/DWG Export Options

- Version.** Select the type supported by the target system.
- Fonts.** Select either **Truetype** or **AutoCAD STANDARD only**.
- Custom Map SolidWorks to DXF.** Select this check box to implement mapping. See **DXF/DWG Mapping** on page 14-14.
- Don't show mapping on each save.** Select this check box to prevent the **SolidWorks to DXF/DWG mapping** dialog box from appearing every time you save a drawing with **Custom Map SolidWorks to DXF** selected. The map file specified in the **Map file** box is used.
- Map file.** To use a previously saved map file, type the file name or browse to the desired file.

## DXF/DWG Mapping

You can map selected SolidWorks entities or colors to different layers or colors in a **.dxf** or **.dwg** file. You create map files in the SolidWorks software to assign selected items to designated layers or colors in the **.dxf** or **.dwg** file.

### *To create a map file:*

- 1 With a SolidWorks drawing document open, click **File, Save As**.
- 2 Select **Dxf Files (\*.dxf)** or **Dwg Files (\*.dwg)** in the **Save as type** list.
- 3 Click **Options**.
  - a) Select the **Custom Map SolidWorks to DXF** check box.
  - b) Click to clear the **Don't show mapping on each save** check box.
  - c) Click **OK** to close the **DXF/DWG Export Options** dialog box.
- 4 Type a name for the drawing file in the **File name** box, then click **Save**.

The **SolidWorks To DXF/DWG mapping** dialog box appears.
- 5 Define the layers then map the entities, or map the colors you want to map in the **.dxf** or **.dwg** file.
- 6 Click **OK** to save the SolidWorks drawing with mapping applied to the document.

### *To save a map file:*

- 1 Click **Map File Settings** in the **SolidWorks To DXF/DWG Mapping** dialog box.
- 2 Select the **Save map file** check box, and clear the **Load map file** check box.
- 3 Type a path and name for a new map file in the box, or browse to an existing file.
- 4 Click **OK**.

**To use a previously saved map file:**

- 1 Click **Map File Settings** in the **SolidWorks To DXF/DWG Mapping** dialog box.
- 2 Select the **Load map file** check box, and clear the **Save map file** check box.
- 3 Browse to an existing file, and click **OK**.

---

**NOTE:** Previously saved map files can contain information for each of the mapping tabs. Review all tabs prior to saving the SolidWorks drawing document.

---

## Define Layers

The **Define Layers** tab in the **SolidWorks To DXF/DWG Mapping** dialog box displays the existing layers. You can add, edit, or delete layers for your mapping file. Layers **0** and **DEFPOINTS** are fixed layers; you cannot edit or delete them.

**To add a layer:**

- 1 On the **Define Layers** tab, click **Add**.  
The **Add New Layer Definition** dialog box appears.
- 2 Type a name for the new layer in the **Layer** box.
- 3 Select an item from the **DXF Colors** list, the **Line Style** list, or both, then click **OK**.

The **Define Layers** tab displays the new layer.

If you did not select a color or a line style, **Unspecified** appears in the **Define Layers** tab. **Unspecified** means that no mapping exists for either the color or the line style of the entities on the layer.

**To edit a layer:**

- 1 Click the layer in the **Define Layers** tab, then click **Edit**.  
The **Edit Layer Definition** dialog box appears.
- 2 Change the items, then click **OK**.

**To delete a layer:**

Click the layer in the **Define Layers** tab, then click **Delete**.

If a layer is referenced in one or more entity mappings, you cannot delete the layer.

## Map Entities

The **Map Entities** tab in the **SolidWorks To DXF/DWG Mapping** dialog box displays the existing entity mappings. You can add, edit, or delete new entity mappings for your mapping file.

### *To map an entity:*

- 1 On the **Map Entities** tab, click **Add**.

The **Add New Entity Mapping** dialog box appears.

- 2 Select an item from each of the four lists: **Entity**, **DXF Color**, **Line Style**, and **Layer**.
- 3 Click **OK**.

The **Map Entities** tab displays the new entity mapping.

If you did not select an item from the **DXF Color** list or the **Line Style** list, **Bylayer** appears in the **Map Entities** tab. **Bylayer** means that the color and line style of the entity are the same as the properties of the layer.

- 4 Click **OK** to accept the changes and save the drawing file, or click **Cancel** to cancel the save operation, or click **Reset All** to return to the default map settings.

### *To edit an entity mapping:*

- 1 Click the entity in the **Map Entities** tab, then click **Edit**.

The **Edit Entity Mapping** dialog box appears.

- 2 Edit the items, then click **OK**.

### *To delete an entity mapping:*

Click the entity in the **Map Entities** tab, then click **Delete**.

## Color Mapping

The **Color Mapping** tab allows you to map the colors used in SolidWorks drawings with the standard DXF color numbers. Color mapping can either work on its own or with entity mapping. Color mapping definitions take precedence over entity mapping definitions.

### *To map SolidWorks colors to DXF standard colors:*

- 1 On the **Color Mapping** tab, click a box in the **SolidWorks Color** column.
- 2 Click a box in the **DXF Color** column to which you want to assign the SolidWorks color, then click **Add**.

The *SolidWorks color to DXF color* mapping is displayed in the **Summary** column.

- 3 To delete a color mapping, click the mapping in **Summary**, then click **Delete**.
- 4 Click **OK** to accept the changes and save the drawing file, or click **Cancel** to cancel the save operation, or click **Reset All** to return to the default map settings.

## VRML Files (.wrl)

---

VRML files can be used to display 3D graphics over the Internet.

When you export to a VRML file, the software takes the **View Section** option into account. If **View Section** is on, only the visible geometry is exported to the VRML file. Appropriately, this is different from the way the software behaves when you save to an IGES, STEP, Parasolid, or ACIS file.

The import of VRML files attempts to knit the VRML polygons into a solid. There are no contoured surfaces in the imported file; all surfaces are planar. This import option is intended as a last resort for importing data into SolidWorks. In many cases (especially complex models) the data does not knit into a usable solid, and is useful for viewing purposes only.

### *To specify the default export options for VRML assemblies:*

- 1 Click **File, Save As** with an assembly document open.
- 2 Select **VRML (\*.wrl)** in the **Files of type** list.
- 3 Select the **Save all components of an assembly in a single file** check box to include each assembly and sub-assembly component in one file.
- 4 Type a name for the file, and click **Save**.

## TIFF Files (.tif)

---

You can save any SolidWorks document as a TIFF image. You can capture the image as it appears on a normal view (screen) or you can capture the image as it appears on a print preview. For drawing documents, you can select the sheet size and scale factor.

You can import a TIFF image and use it as a background of a part or assembly.

### TIFF Options

- Image Type.** Select either **Black & White (Bilevel)** or **RGB Full Color**.
- Compression Scheme.** Select the type supported by the target system:
  - **Uncompressed.**
  - **Packbits.** Results in the smallest file size for a color image.
  - **Group 4 Fax.** Results in the smallest file size of the three formats, but can only be used for a black and white image.

- ❑ **Capture Type.** Select either **Screen capture** or **Print capture**.
  - **Screen capture**, for all SolidWorks documents, captures the image at the resolution of the screen display.
  - **Print capture**, for *part and assembly* documents, captures the image at the resolution of a print preview.
  - **Print capture**, for *drawing* documents, captures the image at a user-defined **DPI** (dots per inch) and **Paper size**.

**To import a TIFF file:**

- 1 In a part or assembly document, click **Insert, Picture**.
- 2 In the **Open** dialog box, browse to the desired file, and click **Open**.

The selected file opens as a background image of your part or assembly.

You can import **RGB Full Color** and **Black & White (Bilevel)** images, and you can also import **Grayscale** and **Palette** images from other systems.

**To turn the display of the TIFF image on or off:**

Click **View, Display, Picture**.

When the **Picture** option is selected, the image is displayed; when the **Picture** option is cleared, the image is hidden.

**To delete a TIFF image from a document:**

Click **View, Modify, Picture, Delete**.

**To replace one TIFF image with another:**

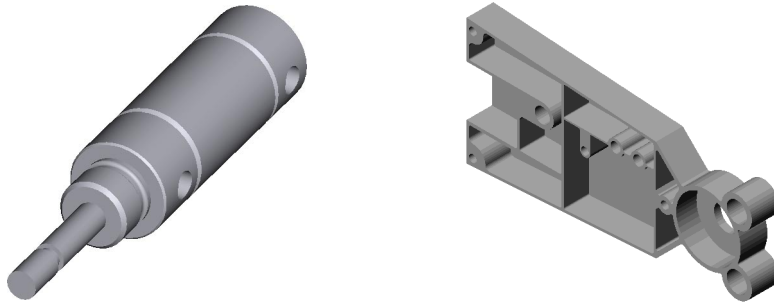
- 1 Click **View, Modify, Picture, Replace**.
- 2 In the **Open** dialog box, browse to the TIFF file that you want to use as a replacement, and click **Open**.



## FeatureWorks

---

The FeatureWorks™ software recognizes a limited set of features on an imported solid in a SolidWorks part document. You can then modify the converted features using standard SolidWorks functionality. The FeatureWorks software is primarily intended for machined parts and sheet metal parts similar to these illustrated models:



- FeatureWorks is fully integrated with the SolidWorks software.** FeatureWorks is supplied as a SolidWorks dynamic link library (.dll) add-in. You access all the controls for the FeatureWorks interface from the FeatureWorks item on the main SolidWorks menu bar. This menu item is displayed whenever a SolidWorks part document is open.
- FeatureWorks recognizes the following types of features:**
  - *Extruded or revolved features*
  - *Chamfers* on linear or circular edges
  - *Constant radius fillets* on linear or circular edges
- Features are recognized on imported parts.** You can recognize features on models imported from IGES, STEP, Parasolid, ACIS, and VDAFS. If you have imported any Unigraphics part files using the SldUG Add-In, or Pro/ENGINEER part files using the SldTRANS Add-In, you can also recognize features on these part files.
- Features recognized by FeatureWorks are fully editable.** Recognized features are the same as features that you create using the SolidWorks software. You can edit the definition of recognized features to change their parameters. For features that are based on sketches, you can edit the sketches to change the geometry of the features.

### **To turn on the FeatureWorks software:**

- 1 Click **Tools, Add-Ins**.
- 2 Select the **FeatureWorks** check box, and click **OK**.

## FeatureWorks Options

*To specify default options for feature recognition:*

- 1 Click **FeatureWorks, Options**.
- 2 In the **Options** dialog box, select from the following options, then click **OK**.

**Feature recognition** (during Automatic Feature Recognition)

- **Basic features.** Select this check box to include extrusions and revolves.
- **Chamfers and fillets.** Select this check box to include chamfers and fillets.

**Part document**

- **Overwrite existing part.** Create the new features in the existing part document, and replace the original imported body.
- **Create new part.** Create the new features in a new part document.

**Prompt for feature recognition as part opens**

When selected, feature recognition begins automatically when you open a part as an imported solid body in a SolidWorks part document from another system.

## Feature Recognition

The FeatureWorks software recognizes features on an imported solid body in a SolidWorks part document. As features are extracted from the imported solid body, the features are listed in the FeatureManager design tree, and they are fully editable within the SolidWorks software. The solid body changes in the graphics area to include only the geometry that either *cannot* be recognized, or has not been recognized yet. If all of the imported geometry is recognized, the imported body feature, **Imported1**, is replaced with fully editable features.

---

**NOTE:** Feature recognition cannot occur if the SolidWorks option **Use fully defined sketches** is selected. Click **Tools, Options, General**, and clear the **Use fully defined sketches** check box, if necessary.

---

*To recognize features:*

- 1 Import a solid body to a part document using IGES, Parasolid, ACIS, STEP, or VDAFS. You can also use the SldUG Add-In to import a Unigraphics part file or the SldTRANS Add-In to import a Pro/ENGINEER part file. **Imported1** must be the only feature in the FeatureManager design tree. For more information on importing a file, see **Importing Documents** on page 14-2.
- 2 Click **FeatureWorks, Recognize Features**.

## Interactive Feature Recognition

The FeatureWorks software allows you to interactively recognize features. You select the feature type and the entities that make up the feature you want to recognize.

In general, it is best to follow these rules when recognizing features interactively:

- Recognize fillets and chamfers first. Interactive recognition of any extrusions or revolves is simpler after any fillets or chamfers are recognized and removed from the geometry.
- Perform Automatic Feature Recognition after you simplify the geometry using Interactive Feature Recognition. If you think the FeatureWorks software can recognize the remaining geometry automatically, click **Automatic** to perform Automatic Feature Recognition.

### *To recognize features interactively:*

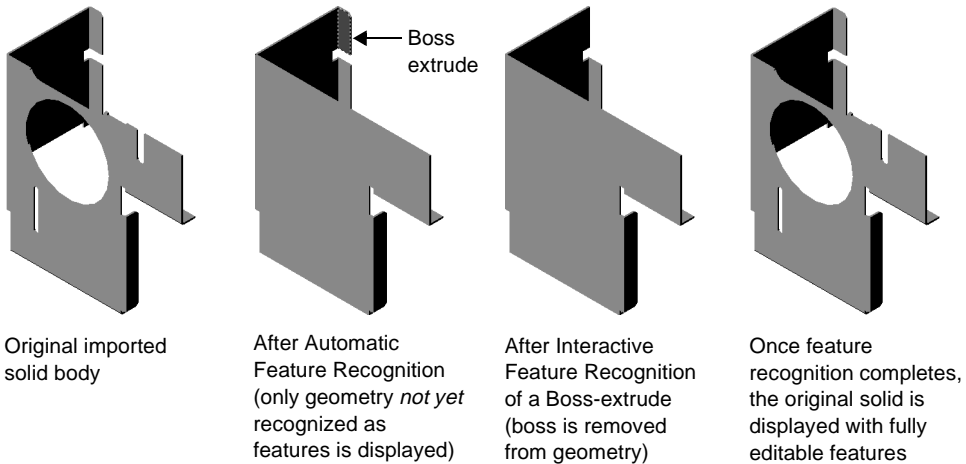
- 1 In the **Recognize Features - Interactive** dialog box, select from these options:
  - **Feature type.** Select the feature type to recognize.
  - **Selected entities.** Select the appropriate face, edges or loop in the graphics area. To determine the appropriate entities for a feature type, see Table 14-2 on page 23.
  - **Up to face.** Select a face for termination of the feature. The FeatureWorks software extends the feature from the sketch plane to the selected face.
  - **Chain revolved faces.** Select this check box if you want the FeatureWorks software to determine the faces for a revolved feature from a minimum set of faces you select. For an illustration of the effect of this check box, see Table 14-2 on page 23.
  - If you select any wrong entities, click **Clear Selections** to clear the **Selected entities** list.
- 2 Click **Recognize Feature** to interactively recognize the selected feature.  
If successful, the feature is *removed* from the imported body, and a new feature is added to the FeatureManager design tree.
- 3 If you want to undo the recognition of a feature, click **Undo**. Click **Redo** to recreate the undone feature.
- 4 If you want to perform Automatic Feature Recognition:
  - Click **Automatic** to perform Automatic Feature Recognition on the current model. See **Automatic Feature Recognition** in the next section for more information.
  - Click **Automatic Options** to open the **Automatic Options** dialog box. Change the desired options.
- 5 Click **Finish** to return to the SolidWorks interface, or click **Cancel** to return to the original imported solid body.

## Automatic Feature Recognition

The FeatureWorks software attempts to automatically recognize and highlight as many features as possible.

- ❑ When the entire imported solid is automatically recognized as features, **Imported1** is replaced by the newly created features in the FeatureManager design tree.
- ❑ When the entire imported solid is *not* automatically recognized as features, click **Continue** to return to the **Interactive Feature Recognition** dialog box, or click **Cancel** to return to the original imported solid body.

The following illustrates the use of both Automatic Feature Recognition and Interactive Feature Recognition on an imported solid body.



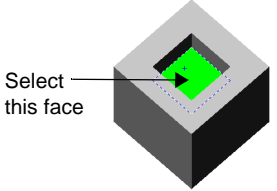
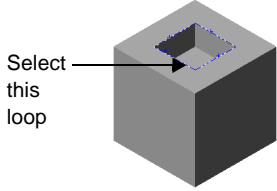
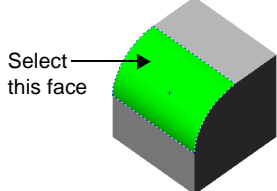
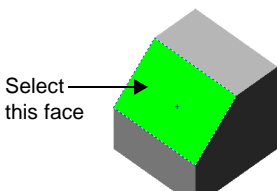
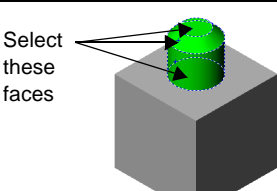
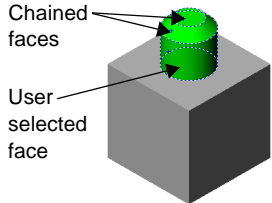
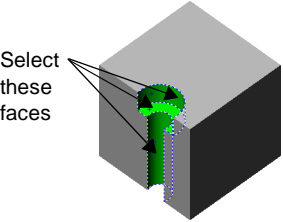
<b>Table 14-2. Interactive Feature Recognition</b>			
<b>Feature type</b>	<b>Selected entities</b>	<b>Required selection</b>	<b>Example</b>
<b>Boss Extrude</b> or <b>Cut Extrude</b>	Face	Select a model face representing the sketch for the feature.	
	Edges or Loop	Select the complete set of edges, or the loop, representing the sketch for the feature.	
<b>Fillet/Round</b>	Face	Select a model face representing the filleted face.	
<b>Chamfer</b>	Face	Select a model face representing the chamfered face.	
<b>Boss Revolve</b> or <b>Cut Revolve</b>	Face	Select the complete set of faces representing the sketch for the revolve feature.	

Table 14-2. Interactive Feature Recognition			
Feature type	Selected entities	Required selection	Example
<b>Boss Revolve</b> or <b>Cut Revolve</b>	Face	If the <b>Chain revolved faces</b> check box is selected, select a single face for this revolve. The FeatureWorks software selects the chained faces.	 <p>Chained faces</p> <p>User selected face</p>
<b>Hole</b>	Face	Select a hole type from the list, then select the complete set of faces representing the sketch for the hole feature.	 <p>Select these faces</p>

## SolidWorks 99 Options

---

This appendix lists all the options available for customizing part, assembly, and drawing documents. To access these options, click **Tools, Options** and select one of the tabs. Change the settings as needed, then either click another tab, or click one of the following:

- **OK.** Accept the changes and exit the dialog box. If you made changes on more than one tab, *all* the changes are accepted.
- **Cancel.** Discard the changes and exit the dialog box. If you made changes on more than one tab, *all* the changes are discarded.
- **Reset All.** Return to the installed system defaults. This restores the original values for *all* the tabs, whether you changed them in this session or in an earlier session.

You specify some options at the *system level*. Options in these categories control certain system behaviors, and they apply to *all* documents:

- General
- Performance
- Drawings
- Edges
- External References
- Reference Geometry
- Routing

You specify other options at the *document level*. You can choose default settings for all new documents, and you can change the values in any individual document, for options in these categories:

- Color
- Grid/Units
- Crosshatch
- Detailing
- Line Font
- Material Properties

## Apply to

When you change document level options, you also need to select an option from the **Apply to** list to specify the scope of the changes:

- **Active Document.** Apply the settings to *only* the currently active document.
- **System Defaults.** Apply the settings to all *new* documents. The currently active document does not change.
- **All Possible.** Apply the settings to both the **System Defaults** and the **Active Document**.

## View System Defaults

To review the system default settings on any of the tabs, click **View System Defaults**. When selected, the system default for each of the options is displayed; when cleared, the option selections used by the active document are displayed.

## Color Options

---

Specifies the color display of view modes, features, and lines. Click the **Edit** button to change the color used to display the selected items. You can select a color from the standard color palette or create a custom color. A preview box shows the current color for the selected item.

## System

In part and assembly documents, the **System** box lists the view modes for color selection: **Wireframe/HLR**, **Shading**, or **Hidden**. In drawings, additional items, such as lines, annotations, and so on, also appear in the box. Select a view mode (or other item) to edit its color representation.

- Apply same color to wireframe/HLR and shaded.** Select if you want to use the same color for these view modes.

## Features

The **Features** box lists the types of features and surfaces to which you can apply a color change.

- Ignore feature colors.** Specifies that the assigned feature colors are not used in the display.
- Reset All.** Resets **Color** tab items only. This is the button at the top right; the **Reset All** button at the bottom left restores the values for *all* the option tabs.
- Edit.** Click the system or feature item to edit, then click **Edit**. The color palette appears. Select a color or specify a custom color.



- ❑ **Advanced.** If you select the **Shading** mode, you can click **Advanced** and use the slider controls to change the display properties:
  - Ambient** – light reflected and scattered by other objects
  - Diffuse** – light scattered equally in all directions on the surface
  - Specularity** – ability to reflect light from a surface
  - Shininess** – a glossy, highly reflective surface
  - Transparency** – ability to pass light through the surface
  - Emission** – ability to project light from the surface
- ❑ **Curvature.** You can display a part or assembly with the surfaces rendered in different colors according to the local radius of curvature (1/radius) in current model units. Click **Curvature** to see the color scale. You can modify the values at five intervals on the color scale; values for the intermediate colors in the range are interpolated. Select the curvature value edit boxes that you want to change and type new values. The defaults are from 0.0010 to 1.0000. The values are not applied to new documents; each new model opens with the default values.
- ❑ **Reset Feature.** Restores the default color setting for the selected feature.
- ❑ **View system defaults.** If the **System** box does not already display a list of lines, borders, dimensions, temporary graphics, and so on, select **view System Defaults**. Now, depending on the type of document that is active, you can select from a large list of items and view modes for which you can change the color. Click **Edit** to select a color for a selected item.

See page A-2 for information about the **Apply to** list.

## Crosshatch Options

---

Specify a crosshatch pattern to be used in section views and in user-defined hatched areas.

See also **Modifying the crosshatch** on page 9-23, and **Area Hatch** on page 10-40.

### Type

Displays the currently selected pattern.

### Properties

- **Pattern.** Select from the list a pattern that is appropriate for the part material.
- **Scale.** Specify the scale used for the pattern.
- **Angle.** Specify the angle used for the pattern.

## Apply to

Crosshatch options are different than other options in the way that they are applied. You should specify the crosshatch pattern in any *part* document for which you will create a section view. When you use the part in an assembly, the appropriate crosshatch pattern for each component is used in assembly section views also. That way, you do not have to edit the crosshatch pattern on the section faces for each component or region in the section view.

If you change the crosshatch options while an *assembly* or *drawing* document is active, you cannot choose an **Apply to** option. The changes you make are applied to the **System Defaults** (for new documents).

## Detailing Options

---

Set options for detailing and dimensioning in parts, assemblies, and drawings.

### Dimensioning standard

- Specifies the standard to use: **ANSI, ISO, DIN, JIS, BSI, or GOST**.
- **Dual dimensions display.** Specifies that dimensions are displayed in both primary and alternate units, and specifies the placement of the alternate dimension, either **On top** or **On right** (on the same line).
- **Fixed size weld symbols.** When cleared (default), the size of the weld symbol is scaled according to the **Dim Font** size and changes if the **Dim Font** size changes. When selected, the size of weld symbols is dependent on the selected dimensioning standard and remains constant regardless of changes to the **Dim Font** size. This option is unavailable for the GOST dimensioning standard.
- **Display datums per 1982.** Click this check box to use the 1982 standard for the display of datums (available only if you use the ANSI dimensioning standard).
- **Trailing zeroes.** Specifies the display of trailing zeroes. Tolerances are not affected.
  - Smart** – Trailing zeroes are trimmed for whole metric values. (Conforms to ANSI and ISO standards.)
  - Show** – Displays trailing zeroes up to the number of decimal places specified in **Tools, Options, Grid/Units**.
  - Remove** – All trailing zeroes are removed.
- **Alternate section display.** Displays an alternate style of section line (available with ANSI standard only). When selected, the section line does not cross the drawing view; the leaders stop at the ends of the section cut.
 

A ←	A ←
A ←	A ←
Alternate	Normal
- **Centerline extension.** Controls the length of the centerline extension beyond the geometry in a drawing view, when the section cuts through the center of a hole, a circular extruded boss or cut, or a revolved feature. The default value is set

according to standards, but you can modify the value in the edit box. The new centerline length is used the next time the section is rebuilt.

## Notes

- **Note Font.** Specifies the font type and size used for notes.
- **Balloons.** Specifies the default balloon style, size, and text for **Notes**, **BOM balloons** (bill of materials), and **BOM balloon text**.
- **Leader anchor.** Specifies to which side of the text the leader attaches: **Left**, **Right**, or the side **Closest** to the attach point.
- **Text justification.** Select **Left**, **Right**, or **Center**.
- **Display notes with bent leader.** When selected, notes are displayed with a bent leader.
- **Bent leader length.** Specifies the distance between the leader bend and the text of the note.

## Break lines

**Break gap.** Specifies the distance between break lines in a broken view.

## Center marks

- **Size.** Specifies the size of **Center marks**, used with arcs and circles in drawings.
- **Show lines.** Specifies whether the center mark lines are displayed.

## Witness lines

Sets the **Gap** (distance from model) and **Extension** (beyond the arrows) of witness lines.

## Datum feature symbols

Specifies the letter of the alphabet that is used for the next Datum Feature Symbol. The default letter is "A." Successive labels are in alphabetic order.

## Dimensions


- **Dim Font.** Lets you specify the font type, style, and size used for dimensions.
- **Tolerance.** Lets you specify options for display of tolerances.  
**Tolerance display.** Select from these types: **None**, **Basic**, **Bilateral**, **Limit**, **Symmetric**, **MIN**, **MAX**, or **Fit**. Depending on the type you choose, specify tolerance values (**Minimum** and **Maximum variation**). You can specify tolerance display options for **Linear** and **Angular** dimensions separately.  
**Font.** Select the font to use for the dimension tolerance text (not available for all types). By default, the tolerance is displayed in the same font as the dimension. To change the size of the tolerance text, clear the **Use dimension's font** check box, then select either **Font Scale** (type a number from 0 to 10.0) or **Font Height** (type a value).

- **Precision.** Sets the precision (number of decimal places) for dimension values and for tolerances for **Primary units**, **Angular units**, and **Alternate units**.
- **Add parentheses by default.** Specifies that reference dimensions in drawings are displayed within parentheses.
- **Use system separator.** Specifies that the default system decimal separator is used in the display of decimal numbers. (To set the system default use **Control Panel**, **Regional Settings** (or **International**), **Number Format**, **Decimal Symbol**) To set a decimal separator different from the system default, click to clear and enter the symbol that you want to use (usually the period or the comma).
- **Snap text to grid.** Specifies that the placement of dimension text snaps to the grid in a drawing or a sketch.
- **Center text.** Specifies that the dimension text is centered between its witness lines.

## Virtual Sharp

Represents a sketch point at the virtual intersection point of two entities. Dimensions and relations to the virtual intersection point are retained even if the actual intersection no longer exists, such as when a corner is removed by a fillet. Select one style of virtual sharp from the list displayed graphically. (This applies to both drawings and sketches.)

## Arrows

- **Size.** Sets **Height** and **Width** of dimension arrowheads, and **Length** of arrow.
- **Section/View Size.** Sets **Height** and **Width** of arrowheads, and **Length** of arrow used in section views.
- **Attachments.** Sets style of arrowheads for attaching annotations to different entities. You can select a different style for attachment to an **Edge/vertex**, to a **Face/surface**, and for **Unattached** annotations.
- **Dimensions.**
  - Style.** Sets the arrowhead **Style** to use for dimensions.
  - Specify placement of arrows in relation to the witness lines: **Outside**, **Inside**, or **Smart**. The **Smart** option changes the placement of the arrows to outside if inside arrows interfere with the text of the dimension.
  - Display 2nd outside arrow (Radial).** Specifies that two outside arrows are displayed with radial dimensions.
  - Arrows follow position of text (Radial)** (only available if **Smart** placement is selected). Allows arrows on radial dimensions to display inside the arc or circle when the text is inside the dimensioned arc or circle, and to move to the outside when the text position moves outside the dimensioned arc or circle.
- **Dimension offset distances.** Lets you specify values for the distances between baseline dimensions. **From model (A)** is the distance between the model and the first baseline dimension. **From last dimension (B)** is the distance between dimension lines. This value is also used for **Align parallel** . See **Baseline Dimensions** on page 10-6 and **Aligning Dimensions** on page 10-9.

## Annotations

- **Display filter.** Specifies which annotation types, if any, are displayed by default. To select specific types, clear the **Display all types** check box and select the types you want.
- **Text scale.** The default is 1:1 relative to the default text size.
- **Always display text at the same size.** When selected, all annotations and dimensions are displayed at the same size, regardless of zoom.
- **Display items only in the view in which they are created.** When selected, any annotation is displayed only when the model is viewed in the same orientation as when the annotation was added.
- **Display annotations.** Toggles the display of annotations.
- **Use assembly's setting for all components.** Sets the display of *all* annotations to match the setting for the assembly document, regardless of the setting in the individual part documents.
- **JIS surface finish size.** If you use JIS dimensioning standard, specify the size (1, 2, or 3 characters) or a custom scale for surface finish symbols.
- **Scale.** If you select **Custom scale** for the **JIS surface finish size**, you specify the scale here. The default is 1:1.

## Leaders

Leaders follow the dimensioning standards by default. To choose a leader style different from the standard, select **Override standard's leader display** and click the style display.

## Section Font, Detail Font

Specify different text fonts for the letter labels on section lines and detail circles, and the corresponding view labels.

See page A-2 for information about the **Apply to** list and the **View System Defaults** option.

## Drawings Options

---

Set default options for drawings.

### Default sheet

**Sheet Scale.** Specifies the default drawing scale for those cases when you choose **No Template** from the **Template to Use** dialog box when opening a new drawing. This scale is used for all views (except **Detail** views) on the sheet. The default is 1:1 of the size on the drawing relative to actual size.

### Type of projection

Specifies either **First angle** or **Third angle** projection for those cases when you choose **No Template** from the **Template to Use** dialog box when opening a new drawing.

### Default display for new drawing views

Specifies the way models or assemblies are displayed in drawings.

- Select the default view mode:
  - Wireframe** – All edges are displayed.
  - Hidden in gray** – Displays visible edges normally; displays hidden edges in gray.
  - Hidden lines removed** – Displays only edges that are visible at the chosen angle; obscured lines are removed.
- If you selected **Hidden in gray** or **Hidden lines removed**, select the mode for viewing tangent edges:
  - Tangent edges visible** – The transition edge between rounded or filleted surfaces displays as a line.
  - Tangent edges with font** – The transition edge between rounded or filleted surfaces displays as a line using the default font for tangent edges defined in **Tools, Options**, on the **Line Font** tab. See **Line Font Options** on page A-17.
  - Tangent edges removed** – The transition edge between rounded or filleted surfaces and other surfaces is not displayed.
- Automatic placement of imported dimensions from model.** When selected, imported dimensions are automatically placed at an appropriate distance from the geometry in the view.
- Display drawing view borders.** When selected (default), displays a border (a gray box) around each drawing view.
- Automatic scaling of 3 view drawings.** When selected, all the views of a **Standard 3 View** are scaled to fit on the drawing sheet, regardless of the paper size selected.

**Open existing drawings with automatic view update off.**

When *selected*, the software does not attempt to update the views while opening an existing drawing. If the model has changed since the last drawing update, any out-of-date drawing views are marked with a gray crosshatch pattern.

This allows you to open drawings, especially large or complex drawings, more quickly. If **Automatic view update** mode was ON when the drawing was last saved, selecting this option sets the mode to OFF.

When *cleared* (the default), the software updates the views when opening an existing drawing if **Automatic view update** mode was ON when the drawing was last saved.

**Set automatic view update for new drawings**

When *selected* (the default), the **Automatic view update** mode is set to ON in all new drawings. When you change the model, the drawing views update automatically.

When *cleared*, the **Automatic view update** mode is set to OFF in all new drawings, and you must update the views manually. For more information, see **Updating Views** on page 9-29.

**Show contents while dragging drawing view.** When selected, the model is displayed as you drag a view. When cleared, only the view boundary is shown while dragging. (Similar to **Show window contents while dragging** in Microsoft Windows.)

**Dynamic drawing view activation**

- When selected, the view closest to the pointer is activated automatically. To stop the dynamic activation temporarily, you can lock a view or the sheet. Right-click the view or sheet, and select **Lock View Focus** or **Lock Sheet Focus**. Then right-click again, and select **Unlock View Focus** (or double-click another view) or **Unlock Sheet Focus** (or double-click any view) to return to dynamic mode.
- When cleared, you double-click a drawing view to activate it, or right-click the view and select **Activate View**. To activate the sheet, double-click a blank area of the sheet, or choose **Activate Sheet** from the right-mouse menu.

**Select hidden entities.** Allows you to select hidden (removed) tangent edges.

**Automatic update of BOM.** When selected, the bill of materials in an assembly drawing is updated automatically to reflect component changes in the assembly. See **Bill of Materials (BOM)** on page 10-31 for more information.

**Detail view scaling.** Specifies the scaling for detail views. The scale is relative to the default scale of the sheet.

## Detail item snapping

See also **Aligning Annotations** on page 10-19. These options may be used together.

- Infer when dragging corner.** When you select a detail item (note, dimension, etc.) and drag it by the corner handle, the corner can infer to the corners of other detail items.
- Infer when dragging center.** When you select a detail item (note, dimension, etc.) and drag it by the center handle, the center can infer to the centers of other detail items.

## Print out-of-date drawing views with crosshatch

Specifies what happens when you **Print** (or **Print Preview**) a drawing that contains out-of-date views, with **Automatic view update** OFF. (Right-click the drawing name in the FeatureManager design tree to turn **Automatic view update** OFF or ON.)

- **Prompt** (default). Notifies you if the drawing contains out-of-date views, and asks how to proceed. When the dialog box appears, click **Yes** to print the drawing *with* crosshatch on the out-of-date views, or click **No** to print the drawing *without* crosshatch.
- **Always.** The printed drawing always includes crosshatch on out-of-date views.
- **Never.** The printed drawing never includes crosshatch on out-of-date views.

If **Automatic view update** is ON, the views update automatically, and the crosshatch is removed, whenever you print the drawing.

## Edges Options

---

Set options for the display of various edge types.

### Hidden Edges

- **Gray or Dashed.** Specifies whether hidden edges are displayed as gray lines or dashed lines.
- **Select hidden for wireframe/HLG.** Allows you to select hidden edges or vertices in **Wireframe** and **Hidden In Gray** modes.
- **Select hidden for HLR and shaded.** Allows you to select hidden edges or vertices in **Hidden Lines Removed** and **Shaded** modes.

### Part/Assembly Tangent Edge Display

Controls the display of tangent edges in **Hidden Lines Removed** and **Hidden In Gray** modes.

- **Tangent edges visible** – Tangent edges are displayed as solid lines.
- **Tangent edges as phantom** – Tangent edges are displayed using the Phantom style line font (a broken line).
- **Tangent edges removed** – Tangent edges are not displayed.



## Shaded Display

- **No edges** (default). No edges are shown in shaded view mode.
  - **HLR edges**. Any edges that would appear in **Hidden Lines Removed** mode are displayed in **Shaded** mode also.
  - **Wire edges**. All edges are shown in shaded mode (wireframe).
- Repaint after selection in HLR**. When selected, specifies that a selected feature is repainted in **Hidden Lines Removed** mode if you click on space. When cleared, specifies that the selected feature is repainted in wireframe.
- Highlight all of selected feature**. When selected, an entire feature is highlighted when you click on it.
- Dynamic highlight from graphics view**. When selected, model faces, edges, and vertices are highlighted when you move the pointer over a sketch, model, or drawing.

## External References Options

---

Specify how assembly, part, and drawing files with external references are opened and managed.

- Open referenced documents with read-only access**. When selected, all referenced documents are opened for read-only access by default.
- Don't prompt to save read-only referenced documents (discard changes)**. Specifies that when this part is saved or closed, no attempt is made to save its referenced documents that are opened for read-only access.
- Allow multiple contexts for parts when editing in an assembly**. Allows you to create external references to a single part from more than one assembly context. Any individual feature or sketch within the assembly may have only one external reference, however.
- Search document folder list for external references**. Specifies that the document folder list is searched to locate or update any referenced documents. See **Searching for Referenced Documents** on page 7-4 for information about using the document folder list.
- Load referenced documents**. Lets you specify whether to load referenced documents when you open a part that has external references (such as a derived part, or a part that has been edited in the context of an assembly). See also **File Management with External References** on page 5-18).
- **Never**. Referenced documents are never opened.
  - **Always**. Referenced documents are always opened, in a minimized window.
  - **Prompt**. Ask about loading externally referenced documents each time you open a derived part. Click **Yes** or **No** in the dialog box that appears when you open the part. If you also click **Don't ask me again** (before you click **Yes** or **No**), the option is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

## Folders

Specify the search rules for locating referenced documents. Folders are searched in the order in which they are listed in the **Show folders for** list.

**Show folders for.** Displays search paths for files of various types.

Select a file type from the list (**Documents, Palette parts, Palette features, Palette forming tools, and Blocks**), then specify the paths to locate files of the selected type:

- To add a new directory path to the list, click the **Add** button.
- To delete a directory path from the list, select the path and click the **Delete** button.
- To change the order of the list, select a directory path and use the **Move Up** button or the **Move Down** button as needed.

---

**NOTE:** The paths for **Documents** are searched only if **Search document folder list for external references** is selected.

The paths for **Palette parts, Palette features, Palette forming tools, and Blocks** are *always* searched, whether **Search document folder list for external references** is selected or not.

---

## Assemblies

**Automatically generate names for referenced geometry.**

When this option is cleared (the default), you can mate to parts for which you have read-only access because you are using the internal face IDs of the parts. Unless you will use component replacement, leave this option cleared, especially in a multi-user environment.

When this option is selected, you automatically create surface identifiers (for example, Face1, Face2) at the time you mate the part; therefore, you need write access to the part in most cases. Select this option if you intend to do component replacement using the same surface identifiers, remembering that you need write access to the parts you are using. (Rename the corresponding edges and/or faces on the replacement component to match the edge or face names on the original part.)

**Update component names when documents are replaced.**

When this option is selected (the default), the component name in the FeatureManager design tree is the name of the part file. If you replace a component, the name in the tree updates to use the name of the new part file.

When this option is cleared, you can use the **Component Properties** dialog box to assign a component name in the FeatureManager design tree that is different from the filename of the component. If you replace a component, the name in the tree does not change.

## General Options

---

Allows you to customize SolidWorks behavior and set default values.

### Model

- **Input dimension value.** Automatically displays the modify box for input of a dimension value at the time you place the dimension.
- **Single command per pick.** Sketch and dimension tools deselect after each use. (Double-clicking a tool causes it to remain selected.)
- **Show dimension names.** Displays the dimension's name as well as its value.
- **Show errors every rebuild.** If errors are present in the model construction, display an error message each time the model rebuilds.

### FeatureManager design tree

- **Scroll selected item into view.** Specifies that the FeatureManager design tree should automatically scroll to display the text that is related to the selected items in the graphics area.
- **Name feature on creation.** When you create a new feature, the feature's name in the FeatureManager design tree is automatically selected and ready for you to enter a name of your choice.
- **Arrow key navigation.** Lets you use the arrow keys to traverse the FeatureManager design tree, and expand or collapse the design tree and its contents, as follows:
  - Up arrow** – scrolls up the design tree
  - Down Arrow** – scrolls down the design tree
  - Left arrow at top of design tree** – collapses the design tree
  - Right arrow at top of design tree** – expands the design tree
  - Left arrow on an item in the tree** – collapses the item to hide its contents
  - Right arrow on an item in the tree** – expands the item to display its contents, if any
  - Space bar** – selects the item
- **Dynamic highlight.** Specifies that the geometry in the graphics area (edges, faces, planes, axes, etc.) is highlighted when the pointer passes over the item in the FeatureManager design tree.

### View Rotation

- **Arrow Keys.** Set the angle increment for view rotation when you use the arrow keys to rotate the model.
- **Mouse Speed.** Set the speed of the rotation when you use the mouse to rotate the model or assembly component. Move the slider to the left to get finer control and slower rotation.

## Sketch

- **Use fully defined sketches.** Requires sketches to be fully defined before you can use them to create features.
- **Alternate spline creation.** Lets you create splines by clicking on through points instead of dragging out segments.
- **Display arc centerpoints.** Turns the display of arc centerpoints on or off in a sketch.
- **Display entity points.** Specifies the display of sketch segment endpoints as filled circles in a sketch. The color of the circle indicates the status of the sketch entity: Black = Fully defined, Blue = Under defined, Red = Over defined, Green = Selected. (Over defined and dangling points are always displayed, regardless of the **Options** setting.)
- **Display virtual sharps.** Specifies the display of virtual sharp intersections in 3D sketches and in drawings that contain 3D sketches.
- **Infer from model.** When sketching on the face of an extruded part, inferencing lines and the inferencing pointer relate to the lines of the part.
- **Prompt to close sketch.** When selected, if you create a sketch with an open profile that can be closed with the model edges to create a feature, the system displays a dialog box, **Close Sketch with Model Edges?** You can choose the model edges to close the sketch profile and the direction.
- **Create sketch on new part.** When you select **New, Part**, the part window opens with the sketching area and sketch tools immediately available.
- **Override dims on drag.** Lets you override dimensions by dragging sketch entities. The dimension updates after the drag is completed. (Also available from the menu: **Tools, Sketch Tools, Override dims on drag.**)
- **Automatic relations.** Specifies whether geometric relations are automatically created as you add sketch elements. (Also available from the menu: **Tools, Automatic Relations.**)
- **Display plane when shaded.** When you are editing a sketch in **Shaded** mode, this option displays the translucent plane. (By default, models created in SolidWorks 97Plus or earlier have this selected.)

---

**NOTE:** If the display is slow, due to the translucent plane, it may be because of the **Transparency Quality** specified in **Tools, Options, Performance**. With some graphics cards, the display speed improves if you use **Low (Screen door)** transparency.

---

## Over Defining Dimensions

- **Prompt to set driven state.** When selected, specifies that when you add an over defining dimension to a sketch, a dialog box asks you if the dimension should be *driven*. (The default is to ask.)
- **Set driven by default.** When selected, specifies that when you add an over defining dimension to a sketch, the dimension is set to be *driven* by default.

The above two check boxes can be used together or alone, resulting in different behaviors when you add an over defining dimension to a sketch.

## General

- **Open last used documents at startup.** Select either **Always** or **Never**. Select **Always** if you want the convenience of having the documents that you used most recently open automatically when you start SolidWorks.
- **Number of backup copies per document.** Lets you specify the number of backup copies you want to save of part, assembly, or drawing documents. Specify zero for no backups, or up to 10 backups. The name of a backup copy is “Backup (*n*) of original filename and extension.”
- **Save backup copies in directory.** Lets you name a directory to which all backup copies are stored by default. Use the **Browse** button to find the directory to use.
- **Maximize document on open.** When selected, each document opens to its largest size within the SolidWorks window.
- **Save AutoRecover info every (*n*) changes.** The software can automatically save information about your active part, assembly, or drawing documents so you do not lose your work in case of a system crash. Specify the number of changes that should occur before information is automatically saved in the **changes** box.

---

**NOTE:** In a part or assembly document, a *change* is a rebuild or an action that requires a rebuild, such as the addition of a feature. In a drawing document, a *change* is any action, such as changing a dimension, creating a section view, or adding an annotation.

---

To turn off **AutoRecover**, click to clear the check box.

- **Use English language.** If you selected the use of a language other than English during the SolidWorks installation, you can change to English by selecting this check box. Note that you must exit and re-start SolidWorks for this change to take effect.
- **Show thumbnail graphics in Windows Explorer.** When selected, a thumbnail graphic is displayed in Windows Explorer, instead of an icon, for each SolidWorks part or assembly document. The graphic is based on the view orientation of the model when the document was saved. A palette of 16 colors is used; if the model uses a color that is not available, a similar color is substituted. The thumbnail graphics also are used in the **File, Open** dialog box.
- **Use shaded face highlighting.** To turn off the solid color face highlighting (green by default), clear this option. This restores the face highlighting behavior (dashed edges) of SolidWorks 98 and earlier versions. Some third party applications may require that you clear this option.
- **Edit design tables in separate window.** By default, when you edit a design a table or insert a new design table, the worksheet is opened within the SolidWorks window, and the Excel toolbars replace the SolidWorks toolbars while the table is

active. When this option is selected, the worksheet is opened in a separate Excel window instead.

- **Enable sketch PropertyManager.** Displays the PropertyManager tab for entering sketch information by coordinate values. For example, when you sketch a line, you can specify the coordinates of each end, the line length, the angle from the horizontal, and horizontal and vertical distances. The properties available are different for each type of sketch entity.

## Grid/Units Options

---

Sets sketch grid properties and units of measure.

### Grid

#### Properties

- **Display grid.** Turns the sketch grid on or off.
- **Dash.** Toggles between solid and dashed grid lines.
- **Automatic scaling.** Adjusts display of the grid when you zoom in and out.
- **Major grid spacing.** Specifies the space between major grid lines.
- **Minor-lines per major.** Specifies the number of minor grid lines between major lines.

#### Snap Behavior

- **Snap to points.** Turns grid snap on or off. When snap is on, points that you sketch or drag snap to the nearest intersection of grid lines (or to intermediate points, if **Snap points per minor** is greater than 1).
- **Snap points per minor.** Specifies the number of snap points between minor grid lines.
- **Snap to angle.** Specifies the number of degrees between snap points when sketching arcs.
- **Snap only when grid is displayed.** Turns off snap behavior when the grid is not displayed.

### Units

#### Length Unit

- Select from: **Millimeters, Centimeters, Meters, Inches, Feet, or Feet and Inches.**
- If you select **Inches** or **Feet and Inches** as your unit of measure, select **Decimal** or **Fractions.**
- If you select a metric unit, or if you select **Decimal** (not **Fractions**) for an English unit, specify the number of **Decimal Places.**
- If you choose **Fractions**, specify the default **Denominator.** (Only dimensions that are evenly divisible by this denominator are displayed as fractions.)
- If you choose **Fractions** and **Inches** or **Feet & Inches**, you can select **Round to Nearest Fraction.**

### Angular Unit

- Select from: **Degrees**, **Deg/Min**, **Deg/Min/Sec**, or **Radians**.
- If you choose **Degrees** or **Radians**, specify the number of decimal places.

### Spin Box Increments

- **Length**. The number of units added or subtracted when you click on a spin box arrow to change a linear dimension value.
- **Angle**. The number of degrees added or subtracted when you click on a spin box arrow to change an angular dimension value.

See page A-2 for information about the **Apply to** list and the **View System Defaults** option.

## Line Font Options

---

Specifies the style and weight of edge lines for selected kinds of edges in drawings.

### Type of Edge

Select an edge type from a list of line types: **Visible Edges**, **Hidden Edges**, **Sketch Curves**, **Dimensions**, **Section Line**, and so on.

### Line Style

From the list, choose a line style to apply to the previously selected edge type. You can choose line styles such as **Solid**, **Dashed**, **Phantom**, and so on.

### Line Weight

From the list, choose a line weight. Choose from **Normal**, **Thin**, or one of several **Thick** lines. The thickness of the lines corresponds to the **Line Weights** for printing that you specify under **File**, **Page Setup**, on the **Printer** tab.

The **Preview** box displays the selected line.

### Apply to

- If the active document is a *part* or *assembly*, you cannot choose an **Apply to** option. The changes you make are applied to the **System Defaults** (for new drawings).
- If the active document is a drawing, you can specify where the changes are applied. See page A-2 for information about the **Apply to** list.

## Material Properties Options

---

Specifies the material properties for the current part. This option is available when you are working with an active *part* document.

This tab is also used by some **Add-In** applications. Refer to the instructions of the specific application for further information.

### Properties

Specifies material properties by selecting the text in the box and entering the appropriate **Density** specification.

See page A-2 for information about the **Apply to** list and the **View System Defaults** option.

## Performance Options

---

Sets performance options. You can choose the display quality that is best for your needs, understanding that higher display quality affects the speed of redrawing the model.

### Display Quality

#### Shaded

Controls the tessellation of cylindrical surfaces for shaded rendering or Stereolithography (STL) output. A higher resolution setting results in slower model rebuild, but more accurate curves.

- **Coarse** and **Fine** are preset resolutions. **Custom** allows you to specify the resolution by dragging the slider control or by entering a **Deviation** value.
- **Deviation**. Reports the maximum chordal deviation in effect at various tuning levels. The chordal deviation value is greater at the **Coarse** setting, and decreases as the resolution becomes more finely tuned. The range of values for the deviation is relative to the overall size of the part.

#### Wireframe

- **Optimal**. A preset quality that provides a faster redraw without greatly sacrificing the display quality.
- **Custom**. Allows you to choose the display quality by dragging the slider control. Choose the **Low** end of the slider if you want to redraw the screen more quickly and the display quality is not of great importance; choose the **High** end of the slider for high display quality, but a slower redraw.

---

**NOTE:** If you are noticing problems with your **HLR** image display, you may want to select a higher quality wireframe display.

---



## Rebuild

- ❑ **Verification on rebuild** controls the level of error checking when creating or modifying features. For most applications, the default setting (cleared) is adequate and results in faster model rebuild. See also **Custom Properties** on page 5-21.
- ❑ **Dynamic update on sketch drag.** You can see a dynamic feature preview when you drag the entities of a sketch, either with or without opening the sketch itself. This is a fast, easy way to edit any feature that is based on one or more sketches.
  - When selected, the preview updates if you pause momentarily while dragging a sketch entity, or when you modify a dimension. If the model is displayed in **Shaded** mode, the preview is transparent.
  - When cleared, the preview updates when you release the pointer after dragging.

## Windows 95 zooming

When using SolidWorks on Windows 95 or Windows 98, there is a limit beyond which you cannot zoom in on a model. The **Enable clipping for zoom limitation** option lets you select a portion of the model and zoom in on only the selected portion. You can zoom in on small details very closely, but the display is slower.

## Transparency quality.

Select either **Low** or **High**. **Low** quality transparency (Screen Door) is similar to viewing an object through a mesh or screen; **High** quality transparency (Alpha Blending) is similar to looking through clear glass.

## Curvature generation

- When **Only on demand** is selected (the default), the initial curvature display is slower, but it uses less memory.
- When **Always (for every shaded model)** is selected, the curvature displays more quickly on the first display, but extra memory is always used (RAM and disk) for every part that you create or open.

## Assemblies

- ❑ **Automatically load parts lightweight.** You can change this setting only when no assemblies, or drawings of assemblies, are open. While this option is selected, all the individual parts in assemblies that you open are loaded **Lightweight**. Sub-assemblies are not lightweight, but the parts that they contain are lightweight. See **Suppression States** on page 6-40 for more information.
- ❑ **Resolve lightweight parts.** Some operations (such as **Mass Properties**, **Interference Check**, displaying non-shaded views and section views, exporting to other file formats) require certain model data that is not loaded for lightweight parts. This option

controls what happens when you request one of these operations in an assembly that has lightweight parts.

- **Always.** Lightweight parts are automatically resolved.
  - **Prompt** (default). The system offers to resolve lightweight parts for you each time one of these operations is requested. In the dialog box that appears, click **Yes** to resolve the parts and continue, click **Cancel** to cancel the operation. If you also click **Always resolve** (before you click **Yes** or **Cancel**), the option is set to **Always**.
- Rebuild assembly on load.** Lets you specify whether you want assemblies to be rebuilt, to update any changed components, when you open them.
- **Never.** Open assemblies without rebuilding.
  - **Always.** Always rebuild assemblies when opening.
  - **Prompt.** Ask each time an assembly is opened if rebuild is desired. Click **Yes** or **No** in the dialog box that appears when you open the assembly. If you also click **Don't ask me again** (before you click **Yes** or **No**), the option selected under **Rebuild assembly on load** is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

## Reference Geometry Options

---

### Display filters

Sets a filter for the default display of reference geometry items. Select the check boxes beside the types of items to display: **Planes**, **Axes**, **Temporary Axes**, **Origins**, **Coordinate Systems**, and **Points**.

---

**NOTE:** To change the visibility of a type of reference geometry in the *current* document only, use the options under the View menu. To change the visibility of an individual reference geometry item, right-click the item and select **Hide** or **Show**.

---

### Plane default names

Specify default plane names for parts and assemblies. For example, you may want to name planes Front, Top, and Right, instead of Plane1, Plane2, and Plane3. Type new names in the boxes that correspond to the original plane names.

## Routing Options

---

Specifies options related to routing operations. See also Chapter 13, “Piping.”

- Library folder path.** Click **Browse**, then select the folder where the routing components are stored. It is recommended that you store *all* routing components in the same folder. When you create or edit a routing assembly, the software searches in this folder by default for the components to use.

If you want to add fittings by dragging them from the Feature Palette window, be sure to include this folder in the folders list for **Palette parts** on the **External References** tab also. See **External References Options** on page A-11.

- Create custom fittings.** Select this check box if you want the software to create new configurations of elbow fittings when needed. If the elbow part for the current route does not contain a configuration with the required bend angle, a new configuration is created automatically. This occurs only if:
  - A configuration with the required bend radius and diameter exists.
  - The required bend angle (the angle *through which* the material is bent) is less than the default bend angle of the existing configuration.
- Create pipes on open line segments.** Select this check box if you want a pipe to be generated for 3D sketch segments that are connected to a fitting at only one end. For example, if you have not yet added a fitting at the end of the route, the last segment of the sketch is considered an open segment.

If this check box is cleared, no pipe is generated on the open segment.

---

**NOTE:** Pipes are *never* generated for sketch segments that are not connected to a fitting at either end, or for segments that are not part of the pipe sketch.

---

- Automatically create sketch fillets.** Select this check box if you want the software to automatically add a fillet at intersections in the 3D sketch. The fillet radius is determined by the **Bend radius** specified in the **Fabricated Piping** dialog box (displayed when you begin the route). This value driven by the bend angle in the selected configuration of the elbow fitting.

# SolidWorks Application Programming Interface

---

This appendix contains an overview of the SolidWorks Application Programming Interface (API). The SolidWorks API is an OLE programming interface to SolidWorks. The API contains hundreds of functions that can be called from Visual Basic, VBA (Excel, Access, etc.), C, C++, or SolidWorks macro files. These functions give you direct access to SolidWorks functionality, such as creating a line, extruding a boss, or verifying the parameters of a surface.

## Online Documentation

Your SolidWorks CD contains the complete API documentation in an online format. Use it as you would use any conventional Windows online help, accessing topics through the table of contents, the index, or by using the Find capability to search for key words or phrases.

The documentation, which includes a list of the new functionality in this release, is located in the *install\_directory\Samples\Appcomm* folder. Double-click the **API\_help.hlp** document.

The SolidWorks API documentation, API training materials, Frequently Asked Questions, and other useful information can be found on the SolidWorks web page:

**<http://www.solidworks.com/html/Products/api>**

## Topics in this Appendix

- Recording and playing SolidWorks macros
- Using Visual Basic to develop an application
- Programmer's guide

If you need further information, or if you will be developing applications with other languages (such as Visual C++), please refer to the online documentation.

## Recording and Playing SolidWorks Macros



---

You can record operations performed with the SolidWorks user interface and replay them using SolidWorks Macros. These macros contain the equivalent calls to the API functions that were made when the operations were performed with the user interface.

### **To display the Macro toolbar:**

Click **View, Toolbars, Macro**, or right-click in any border of the SolidWorks window, and select **Macro**.

### **To record a macro:**

- 1 Click **Record/Pause Macro**  or **Tools, Macro, Record**.
- 2 Perform the operations you want to have in your macro.
- 3 Click **Stop Macro**  or **Tools, Macro, Stop** and enter a filename for the macro. (Macros have a **.swb** extension.)


---

**NOTE:** Macros recorded in this way usually have some context-dependent elements that require minor changes before they can be used with other documents.


For example, you may need to change references to the part name, so that the macro plays back correctly in another document.

---

### **To run a macro:**

- 1 Click **Run Macro**  or **Tools, Macro, Run**.
- 2 Select the filename (**.swb**) of the macro you want to run, and click **Open**.

### **To edit a macro:**

- 1 Click **Edit Macro**  or **Tools, Macro, Edit**.
- 2 Select the filename (**.swb**) of the macro you want to edit, and click **Open**. You can edit, run, and debug the macro from the macro editor window.

### **To run a macro file from a customized toolbar button:**

- 1 Click **Tools, Customize**, and click the **Macros** tab. (You must have a document open in the SolidWorks window to use the customize functions.)
- 2 Click **Add**. Select the **.swb** file and click **Open**.
- 3 Under **Macro details**, enter a **Name** for the macro.
- 4 Click **Apply**.
- 5 Click the **Commands** tab, and under **Categories**, select **Macro**.
- 6 Drag the toolbar button for the macro from the **Buttons** box, and drop it on the Macro toolbar in the SolidWorks window.
- 7 To run the macro, click the button you just added.

## Using Visual Basic to Develop an Application

---

Installing SolidWorks to compile and run a Visual Basic application requires no special steps. Install the SolidWorks software according to the instructions provided with the CD.

SolidWorks supports programs through any OLE controller (Visual Basic, Visual Basic for Applications, and so on).

Applications written in Visual Basic can be run in several ways. Whichever way you choose to run the application, the code you generate should be similar to code generated by the SolidWorks Macro utility. To get started, it may be helpful to generate a macro from within SolidWorks (see **Recording and Playing SolidWorks Macros** on page B-2), then use that code as the foundation for your application.


### ***For a program to run from SolidWorks:***

- 1 If your routine uses only SolidWorks API calls and standard Visual Basic 3.0 commands, you can create your program in Visual Basic, and use a filename extension of **.swb** instead of **.bas**. There is no need to compile your application.

---

**NOTE:** SolidWorks Macro files (**.swb**) only recognize Visual Basic 3.0 commands. If you want to use Basic commands in Visual Basic 4.0 or later, you must compile and run the program as a Visual Basic executable or from a VBA application.

---

- 2 Click **Run Macro** , or **Tools, Macro, Run**.
  - 3 Select the desired BASIC source file, and click **Open**.
- You can also customize a SolidWorks toolbar button to run the macro. See the previous section for more information.

### ***For a program to run as a separate .exe file:***

- 1 Build your project with a standard utility such as Microsoft Visual Basic.
- 2 From the Microsoft Visual Basic application, select **File, Make EXE File**.
- 3 Execute the **.exe** file.

If a SolidWorks session is already running, the standard Visual Basic commands `CreateObject` or `GetObject` allow your program to attach to the session.

If a SolidWorks session is not already running, the `CreateObject` command starts a new SolidWorks session.

### ***For a program to run as a DLL:***

SolidWorks does not support Visual Basic DLL implementations.

***For a program to run from another application:***

- 1 Load the application (Access, Excel, etc.)
- 2 Use the embedded VBA to generate your utility or script.
- 3 Use the compile utilities within the application to build your project.
- 4 Load the application, and run your program or script from the application.

If a SolidWorks session is already running, the standard Visual Basic commands `CreateObject` or `GetObject` allow your program to attach to the session.

If a SolidWorks session is not already running, the `CreateObject` command starts a new SolidWorks session.

## Programmer's Guide

---

### Using Macros as a Starting Point

To get a head start on any project, it may be helpful to generate a macro from within the SolidWorks program (see **Recording and Playing SolidWorks Macros** on page B-2). By recording a macro and performing the desired function interactively, you can get a start on the commands and syntax needed for your code.

Record your macro before you do any coding, then use it as a foundation for your project. When you require additional functionality in your program, go back to the SolidWorks program and record additional macros. Cutting and pasting your macros into the existing sample projects can be beneficial for even the most advanced programmer.

### Sample Projects

SolidWorks provides you with several sample applications to get started. Each project directory includes a **readme.txt** file to help you become familiar with each project and its highlighted functionality. In most cases you need to recompile each project on your system before attempting to run it.

Sample projects for both Visual Basic and C++ can be found in the **install\_directory\Samples** folder, in the following folders: **comuserdll**, **userdll**, **TestApp**, and **VisualBasic**. In addition, Visual C++ programmers can use a custom AppWizard utility to create a new Visual C++ project. The AppWizard can be downloaded from <http://www.solidworks.com/html/Products/api/>.

For C++ applications, you can use either a COM or a Dispatch interface. COM is recommended for all add-in DLL projects. Dispatch is required for all **.exe** implementations.

A sample project for each interface type can be found in the **comuserdll** and **userdll** folders respectively. The project in the **TestApp** folder is a second Dispatch example. Refer to the online documentation **install\_directory\Samples\Appcomm** folder (double-click the **API\_help.hlp** document).

Many developers simply load the project file from the appropriate directory, then cut and paste their code into the sample project. You should also be sure to refer to the **.mak** file for appropriate build settings.

Additional samples, training materials, and answers to frequently asked questions can be found on the SolidWorks web site:

**<http://www.solidworks.com/html/Products/api>**

In addition, example programs created by SolidWorks developers and customers can be found in the Subscription Services section of the SolidWorks web site.

---

**NOTE:** The sample projects are supplied on an as-is basis, and are intended to demonstrate the method of using the OLE capabilities of SolidWorks. SolidWorks Corp. makes no representations or warranties regarding these samples.

Any licensed user of SolidWorks is free to use any or all of these samples in connection with building applications related to SolidWorks, and is granted a royalty free, non-exclusive license for these samples, or parts thereof. Intellectual property rights of the samples remain with SolidWorks. Any confidentiality provisions of the SolidWorks license apply to the samples.

---

## SolidWorks API Objects

The SolidWorks API interface uses an object-oriented approach. All of the functions are methods or properties that apply to an object. Thus, there is an assumption that these methods apply to the current state.

For example, there is a method `EditDelete` with effectively no arguments. Instead of passing in the items to delete, the `EditDelete` method acts on the current set of selected items. This style of interface may be different from what you have encountered in previous products. However, it is consistent with the Windows-based approach of SolidWorks. By way of reference, all of the SolidWorks user interface is based on Microsoft Foundation Classes.

To call any of the methods or properties that are found in the API, you first need to obtain the object. For example, to use the `Face::GetArea` method you need to have a `Face` object.

There are many different ways to access objects within the SolidWorks API. For example, to obtain the `Face` object mentioned above, you could either:

- Obtain the `Body` object using `PartDoc::Body` and then traverse the faces on the body using the `Body::GetFirstFace` and `Face::GetNextFace` methods.
- Get the `Face` object from the current set of selected items using `SelectionMgr::GetSelectedObject3`.
- Get the `Face` object by its name using `PartDoc::GetEntityByName`.



## Appendix B SolidWorks Application Programming Interface

Once you have the Face object, you can access the properties and methods found in the Face Class. For example, you could get the number of edges on the face using the `Face::GetEdgeCount` method, or you could get the normal vector for the face using the `Face::Normal` property.

Be aware that some API methods and properties, such as `AssemblyDoc::OpenCompFile`, also require a selected item. To call `OpenCompFile`, you need to have the `AssemblyDoc` object AND have a component selected. The component can be selected in one of two ways:

- The user can interactively select the component.
- You can programmatically select the component using `ModelDoc::SelectByID`.

For more information on object relationships, please refer to the API Object Diagram in the API documentation. Go to the **install\_directory\Samples\Appcomm** folder, and double-click the **API\_help.hlp** document.

# SolidWorks Corporation License and Subscription Service Agreement

---

For general reference only, licenses vary modestly from country to country and your country-specific license may vary somewhat from these terms.

**IMPORTANT – READ CAREFULLY:** The terms and conditions that follow set forth a legal agreement between you (either an individual or an entity), the end user, and SolidWorks Corporation, a Delaware corporation with its principal place of business at 300 Baker Avenue, Concord, Massachusetts 01742 USA (“SolidWorks”) relating to the computer software known as SolidWorks® and certain other related software modules, if applicable (the “Software”). The term “Software” includes and these terms and conditions also apply to, any updates or upgrades to the Software that you may receive from time to time under a subscription service or other support arrangement. You should carefully read these terms and conditions BEFORE opening the case that contains the Software or installing and using the Software. Opening the case containing the Software or installing and using the Software will signify your agreement to be bound by these terms and conditions. If you do not agree to these terms and conditions, promptly return the case containing the Software and the accompanying items (including written materials) for a refund. This is a license agreement and not an agreement for sale.

**1. Grant of License.** SolidWorks grants to you a nonexclusive license to use the Software and the printed and/or electronic user documentation (the “Documentation”) accompanying the Software in accordance with this Agreement. If you have paid the license fee for a single user license, this Agreement permits you to use one copy of the Software on any single computer, provided that the Software is in use on only one computer at a time. If you have paid the license fees for multiple licenses of the Software, then at any time you may have as many copies of the Software in use as you have licenses. The Software is “in use” on a computer when it is loaded into the temporary memory (i.e. RAM). If the potential number of users of the software exceeds the number of licenses you have purchased, then you must have a reasonable mechanism or process in place to assure that the number of computers on which the software is running concurrently does not exceed the number of licenses purchased. At the time of registration (see Section 9 below) you must inform us of the maximum number of potential users of the licenses you purchase. We recommend you also inform us of the names of all potential users so that we can notify them of upcoming updates and other pertinent information. If the software is permanently installed on the hard disk or other storage device of a computer (other than a network server; this sentence does not apply to a network license) and one person uses that computer more than 80% of the time it is in use, then that person may also use the Software on a portable or home

## Appendix C SolidWorks Corporation License and Subscription Service Agreement

computer. You will keep accurate and up-to-date records of the numbers and locations of all copies of the Software, will supervise and control the use of the Software in accordance with the terms of this Agreement, and will provide copies of such records to SolidWorks upon reasonable request.

SolidWorks reserves the right to embed a software security mechanism within the Software to monitor usage of the software to verify your compliance with this license. Such a security mechanism may store data relating to the use of the Software and the number of times it has been copied. SolidWorks reserves the right to use a hardware lock device, license administration software, and/or a license authorization key to control access to the Software. You may not take any steps to avoid or defeat the purpose of any such measures. Use of any Software without any required lock device or authorization key is prohibited.

**2. Ownership of the Software/Restrictions on Copying.** SolidWorks or its licensors own and will retain all copyright, trade secret and other proprietary rights in and to the Software and the Documentation. THE SOFTWARE AND THE DOCUMENTATION ARE PROTECTED BY COPYRIGHT LAWS AND OTHER INTELLECTUAL PROPERTY LAWS. You obtain only such rights as are specifically provided in this Agreement. You may copy the Software into any machine-readable form for back-up purposes and within the license restrictions of Section 1. You may not remove from the Software or Documentation any copyright or other proprietary rights notice or any disclaimer, and you shall reproduce on all copies of the Software made in accordance with this Agreement, all such notices and disclaimers.

**3. Other Restrictions on Use.** This Agreement is your proof of license to exercise the rights granted herein and must be retained by you. You may not use any portion of the Software separately from or independently of the Software and other than for your normal business purposes. You may not sell, license, sublicense, transfer, assign, lease or rent the Software or the license granted by this Agreement. You may not modify or make works derivative of the Software and you may not analyze for purposes competitive to SolidWorks, reverse engineer, decompile, disassemble or otherwise attempt to discover the source code of the Software, except in accordance with Article 8 below, if applicable, as it contains trade secrets of SolidWorks and its licensors.

**4. Subscription Service.** If you subscribe to subscription service for the Software you have licensed hereunder by paying the fee therefor, you will be entitled to receive for such copy: 24 hour by 7 day/ week on-line web access to “down-load” the latest updates to the Software; all major upgrades for the Software released during the subscription period; and telephone support services from an Authorized SolidWorks Reseller. The term of this service runs for one year from the first day of the month following the date you ordered subscription service. It shall automatically renew from year to year unless one party notifies the other party in writing of its desire not to renew the term, at least 30 days prior to the end of a term. Software that is delivered as an upgrade or update to a previous version of the licensed Software must replace the previous version – no additional license is granted; you may install only such number of updates as equal the number of subscription service fees you have paid.

**5. Term.** The license granted herein will continue until it is terminated in accordance with this Article 5. SolidWorks may terminate the license granted herein immediately upon written notice to you (i) for justified cause, including without limitation if you breach any provision of Article 2 or 3 of this Agreement, or (ii) if you breach any other provision of this Agreement and fail to cure such breach within fifteen (15) days of notice thereof. Upon the termination of the license, you will promptly return to SolidWorks or destroy all copies of the Software and Documentation covered by the license as instructed by SolidWorks. The provisions of Articles 2, 3, 5, 7 and 10 of this Agreement shall survive any termination of this Agreement.

**6. Responsibility for Selection and Use of Software.** You are responsible for the supervision, management and control of the use of the Software, and output of the Software, including, but not limited to: (1) selection of the Software to achieve your intended results; (2) determining the appropriate uses of the Software and the output of the Software in your business; (3) establishing adequate independent procedures for testing the accuracy of the Software and any output; and (4) establishing adequate backup to prevent the loss of data in the event of a Software malfunction.

## **7. Limited Warranty, Exceptions & Disclaimers**

**a. Limited Warranty.** SolidWorks warrants that the Software will be free of defects in materials and workmanship and will perform substantially in accordance with the Documentation for a period of ninety (90) days from the date of receipt by you. SolidWorks also warrants that any services it provides from time to time will be performed in a workmanlike manner in accordance with reasonable commercial practice. SolidWorks' entire liability and your sole remedy under this warranty shall be to use reasonable efforts to repair or replace the nonconforming media or Software or re-perform the service. If such effort fails, SolidWorks or SolidWorks' distributor or reseller shall (i) refund the price you paid for the Software upon return of the nonconforming Software and a copy of your receipt or the price you paid for the service, as appropriate, or (ii) provide such other remedy as may be required by law. Any replacement Software will be warranted for the remainder of the original warranty period or thirty (30) days from the date of receipt by you, whichever is longer.

**b. Exceptions.** SolidWorks' limited warranty is void if breach of the warranty has resulted from (i) accident, corruption, misuse or neglect of the Software; (ii) acts or omissions by someone other than SolidWorks; (iii) combination of the Software with products, material or software not provided by SolidWorks or not intended for combination with the Software; or (iv) failure by you to incorporate and use all updates to the Software available from SolidWorks. SolidWorks does not warrant that the Software or service will meet your requirements or that the operation of the Software will be uninterrupted or error free.

**c. Limitations on Warranties.** The express warranty set forth in this Article 7 is the only warranty given by SolidWorks with respect to the Software and Documentation furnished hereunder and any service supplied from time to time; SolidWorks and its licensors, to the maximum extent permitted by applicable law, make no other warranties, express, implied or arising by custom or trade usage, and specifically disclaim the warranties of merchantability and fitness for a particular purpose. In no event may you bring any claim, action or proceeding arising out of the warranty set forth in this Section 7 more than one year after the date on which the breach of warranty occurred.

**d. Limitations on Liability.** Except as required under local law, the liability of SolidWorks and its licensors, whether in contract, tort (including negligence) or otherwise, arising out of or in connection with the Software or Documentation furnished hereunder and any service supplied from time to time shall not exceed the license fee you paid for the Software or any fee you paid for the service. In no event shall SolidWorks or its licensors be liable for special, indirect, incidental, punitive or consequential damages (including without limitation damages resulting from loss of use, loss of data, loss of profits, loss of goodwill or loss of business) arising out of or in connection with the use of or inability to use the Software or Documentation furnished hereunder and any service supplied from time to time, even if SolidWorks or its licensors have been advised of the possibility of such damages. Notwithstanding the foregoing, in no event shall SolidWorks' liability be limited in the case of death or personal injury arising as a result of the negligence or willful misconduct of SolidWorks.

**8. European Software Directive.** If the provisions of the Council of European Communities Directive of May 14, 1991 on the Legal Protection of Computer Programs as implemented in applicable national legislation (the “Software Directive”) apply to your use of the Software, and you wish to obtain the information necessary to achieve interoperability of an independently created computer program with the Software as permitted under Article 6 of the Software Directive (“Interoperability Information”), you must notify SolidWorks in writing, specifying the nature of the Interoperability Information you need and the purpose for which it will be used. If SolidWorks reasonably determines that you are entitled to such Interoperability Information under the Software Directive, SolidWorks shall, at its option, either (i) provide such Interoperability Information to you, or (ii) allow you to reverse engineer the Software, within the limits and for the purposes prescribed by the Software Directive, solely to the extent indispensable to obtain such Interoperability Information. If SolidWorks elects clause (i), you will provide any information and assistance reasonably requested by SolidWorks to enable SolidWorks to perform clause (i), and SolidWorks may charge you a reasonable fee for making available the requested Interoperability Information, unless such a fee is prohibited under the Software Directive.

**9. Shutdown Feature.** You acknowledge and agree that the Software contains an automatic shutdown feature (the “Shutdown Feature”) which if activated will render the Software inoperable. The Shutdown Feature will automatically activate thirty (30) days after the Software is installed unless an access code that overrides the Shutdown Feature (the “Registration Code”) is entered. To obtain your Registration Code you must register with SolidWorks, either by returning the Registration Card included with the Software or providing SolidWorks electronically the information requested in the installation process. Upon receipt of the registration information, SolidWorks will issue you a Registration Code.

**10. General Provisions.** You acknowledge that the Software and the Documentation may be subject to the export control laws of the United States or the United Kingdom and agree not to export or re-export the Software or the Documentation without the appropriate United States or foreign government licenses and the written approval of SolidWorks and its licensors. This Agreement shall be governed by and construed and enforced in accordance with the substantive laws of The Commonwealth of Massachusetts, without regard to the United Nations Convention on Contracts for the International Sale of Goods and will be deemed a contract under seal. If for any reason a court of competent jurisdiction finds any provision of this Agreement, or a portion thereof, to be unenforceable, that provision shall be enforced to the maximum extent permissible and the remainder of this Agreement shall remain in full force and effect.

**11. 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 252.227-7013(c)(1)(ii)(Rights in Technical Data and Computer Software) and in this Agreement, as applicable. Contractor/Manufacturer: SolidWorks Corporation, 300 Baker Avenue, Concord, Massachusetts 01742.

**You further agree that this Agreement is the complete and exclusive statement of your agreement with SolidWorks relating to the Software and subscription service and supersedes any other agreement, oral or written, or any other communications between you and SolidWorks relating to the Software and subscription service; provided, however, that this Agreement shall not supersede the terms of any signed agreement between you and SolidWorks relating to the Software and subscription service.**

# Index

---

- 2D IGES geometry 14-10
- 2D sketching
  - command line emulator 1-38
- 2D sketching in drawings 9-39
- 3 point arc 2-5, 2-9
- 3D curves, IGES 14-8
- 3D sketch 2-50
  - coordinate system 2-50
  - dimensions 2-50
  - in routing 13-3
  - relations 2-51, 2-52
  - virtual sharps 2-52
- 3D spline 3-9, 3-14
  
- A**
- accelerator keys 1-32
- accessing sketch entity tools 2-5
- accessing sketch tools 2-6
- ACIS files 14-11
  - edit imported document 14-4
  - export 14-6
  - export options 14-11
  - import 14-2
  - import options 14-3
  - importing a surface 14-2
- activate inferencing lines 2-29
- add relations 2-37, 2-38
  
- add-in applications
  - 2D sketching 1-38
  - routing 13-1
- adding
  - components, to assemblies 6-6
  - drawing sheets 9-4
  - features 2-31
  - geometric relations 2-29
  - mating relationship 6-20
  - parts, to assemblies 6-6
  - weld bead 7-27
- advanced component selection 6-47
- advanced lighting conditions A-3
- advanced material properties 5-3
- advanced show/hide components 6-48
- advanced smoothing
  - with lofts 4-16
  - with sweeps 4-12
- align, *See also* mating
  - broken views 9-28
  - drawing views 9-31
  - edge 9-31
  - grid 2-21
  - horizontal 9-30
  - vertical 9-30
- aligned section view 9-22
- aligning
  - annotations 10-19
  - dimensions 10-9

## Index

- alignment condition in mating 6-21
- alternate section arrow display A-4
- ambient light 5-27
- anchor point for BOM 9-6, 10-32
- angle mating 6-19
- angular dimensions 2-34
- angular unit of measure A-17
- annotations
  - aligning 10-19
  - area hatch 10-40
  - assemblies 6-51
  - balloon callout 10-38
  - bill of materials 10-31
  - block 10-41
  - center mark 10-24
  - cosmetic thread 10-25
  - creating 10-19
  - custom symbol 10-41
  - datum feature symbol 10-27
  - datum target symbol 10-28
  - displaying in parts 5-22
  - hole callout 10-24
  - import from model 10-18
  - note 10-20
  - options A-7
  - surface finish symbol 10-26
  - weld symbols 10-39
- API
  - documentation 1-17, B-1
  - getting started with Visual Basic B-3
  - installing with Visual Basic B-3
  - macros as shortcuts B-4
  - online documentation B-1
  - playing macros B-2
  - recording macros B-2
  - sample projects B-4
- application protocols, STEP files 14-11
- arc
  - 3 point 2-9
  - centerpoint 2-5, 2-9
  - centerpoints, displaying A-14
  - dimension true length 2-36
  - tangent 2-5, 2-9
- arcs
  - dimensioning between 2-36
- area hatch 10-40
- arrange document icons 1-15
- arrange toolbar icons 1-17
- array. *See* pattern
- arrow options A-6
- arrows, using to rotate view 1-32
- assemblies
  - adding components to 6-6
  - adding palette parts 11-13
  - annotations 6-51
  - assembly features 7-16
  - backing up referenced files 6-11
  - bottom up design 6-2
  - changing component order 6-33
  - colors 6-49
  - component names 6-13
  - component pattern 6-8, 7-11
  - creating 6-2
    - components within 7-6
  - creating configurations 8-2
  - customizing 6-49
  - deleting components 6-8
  - displaying mates with components 6-5
  - dragging components into 6-6
  - editing components 7-7
  - editing configurations 8-5
  - envelopes 6-45
  - explode steps 6-35
  - exploding 6-35
  - feature scope 7-16
  - FeatureManager design tree 6-3
  - file type 6-3
  - hierarchy 6-5
  - inserting components 6-6
  - interference detection 6-38
  - joining parts in 7-24
  - layout sketch 7-14
  - lightweight components 6-40
  - mategroups 6-19
  - mating components 6-19
  - mating relationships 6-19
  - merging components 7-24
  - mold cavity 7-18
  - opening 6-45, 6-49
    - components within 7-8
    - named configuration 6-45
    - with new references 6-14
    - with simplified parts 8-6
  - overlapping parts, *See* interference detection
  - overview 6-3
  - part instances 6-4

- paths of components 6-10
- positioning components 6-17
- preserve mates when replacing
  - components 6-16
- rebuild when loading 6-9
- referenced file locations 6-14
- relationship codes 6-4
- removing components 6-8
- reordering items 6-32, 7-13
- reorganizing the hierarchy 6-32
- replacing parts 6-12
- resolved components 6-40
- roll back 7-13
- saving 6-10, 7-5
- simplifying 6-39
- sub-assemblies 6-28
- suppressing components 6-40
- time-dependent features 7-9
- toolbar 6-4
- top-down design 6-2, 7-1, 7-14
- verifying part files 6-10
- viewing
  - by dependencies 6-5
  - feature detail 6-5
  - hierarchy 6-5
- assembly feature
  - creating 7-16
  - scope 7-16
- assembly window 1-12
- attach dimensions 14-13
- auto hatching, assembly section views 9-26
- auto relief 12-9
  - editing 12-22
  - rectangular 12-9
    - relief ratio 12-9
    - width and depth 12-9
  - tear 12-9
- auto round 4-6
- autoexplode 6-35
- automatic feature recognition,
  - FeatureWorks 14-22
- automatic inferencing lines 2-29
- automatic mating relationships 6-22
- automatic relations 2-37, A-14
- automatic solve 2-21
- automatic view update 9-29
- AutoRecover option 1-8
- auxiliary view 9-15

- axis 3-6
  - temporary 3-6

## **B**

- background image (TIFF file) 14-18
- backing up an assembly 6-11
- balloon callout 10-38
- base 4-1
  - extrude 4-3
  - from a thickened surface 4-56
  - loft 4-15
  - revolve 4-9
  - sweep 4-10
- base part 5-16
- baseline dimensions 10-6
- bend allowance 12-7
- bend table 12-8
- bends 4-7
- bill of materials 10-31
  - anchor point 9-6, 10-32
  - displaying configurations 8-3
  - editing in Excel 10-33
  - expand parameter in a design table 8-16
  - indented sub-assembly 10-32
  - inserting 10-31
  - properties 10-33
  - saving as Excel file 10-36
  - template 10-31
  - using custom file properties 10-36
- blanked surfaces 14-4
- blind extrusion 4-3
- block 10-41
- borders, view 9-11
- boss 4-1
  - extrude 4-3
  - from a thickened surface 4-56
  - loft 4-15
  - revolve 4-9
  - sweep 4-10
- box select, by dragging 1-24
- break alignment 9-30, 9-31
- break lines
  - horizontal 9-20
  - vertical 9-20
- broken view 9-20

## **C**

- calculator 1-31
- cap ends 4-6



## Index

- cap thickness 4-6
- cascade windows 1-14
- casting parts 7-18
- cavity
  - creating in a mold 7-19
  - scaling factor 7-18
- center dimension text 10-10
- center line loft 4-20
- center mark 10-24
- center of mass 5-25
- centerline 2-5, 2-12
- centerline in section view 9-24
- centerpoint arc 2-5, 2-9
- centerpoint ellipse 2-5, 2-10
- chain dimension 10-15
- chamfer 4-28
- check entity 5-19
- check sketch for feature usage 2-24
- circle 2-8
- circles, dimensioning 2-36
- circular pattern 4-42
  - equal spacing 4-42
  - spacing 4-42
  - total angle 4-42
  - total instances 4-42
- circular step and repeat 2-19
- close all documents 1-15
- close along loft direction 4-16
- close sketch to model 2-23
- coincident items, selecting 1-25
- coincident mating 6-19
- coincident relation 2-39
- collinear dimensions
  - align 10-9
- collinear relation 2-39
- color
  - changing feature or face color 5-2
  - changing part color 5-2
  - options A-2
  - part color in assemblies 6-50
- color mapping 14-16
- command emulator, 2D sketching 1-38
- comment, in a design table 8-15
- common properties 3-8
- complex assemblies 7-13
- complexity of sketches 2-49
- components
  - adding to assemblies 6-6
  - adding to sub-assemblies 6-28
  - advanced selection 6-47
  - advanced show/hide 6-48
  - changing the order 6-33
  - configuration, in design table 8-14
  - creating within assemblies 7-6
  - deleting from an assembly 6-8
  - derive component part 5-17, 7-20
  - detecting interference between 6-38
  - dragging into assemblies 6-6
  - editing within assemblies 7-7
  - fixing 6-17
  - floating 6-17
  - inserting into assemblies 6-6
  - lightweight 6-40
  - moving in assemblies 6-18
  - multiple part configurations 8-7
  - names in FeatureManager design tree 6-13
  - opening within assemblies 7-8
  - pattern in assembly 6-8, 7-11
  - positioning in assemblies 6-17
  - properties 8-7
  - replacing in assemblies 6-12
  - resolved 6-40
  - rotating in assemblies 6-18
  - suppressed 6-40
  - suppression state, in design table 8-14
  - updating names 7-4
- computation, automatic solving 2-21
- concentric
  - mating 6-19
  - relation 2-39
- configurations
  - activating 8-4
  - creating 8-2
  - deleting 8-5
  - displaying in a BOM 8-3
  - editing 8-5
  - of parts in an assembly 8-7
  - properties 8-2
  - properties, advanced 8-3
  - supported items 8-3
  - using a design table 8-8
- conformance classes, STEP files 14-11

- connection point 13-3
  - creating 13-4
  - in a design table 13-5
  - nominal diameter 13-5
  - specification 13-5
- constant radius fillet 4-23
- constrain all 2-4, 2-43
- constraining sketch, *See* add relations
- construction geometry 2-5, 3-6, 3-8
  - axes 3-6
  - centerline 2-5
  - planes 3-2
  - point 2-5
  - projected curve 3-9
- construction lines 2-5
- construction planes, setting up 3-2
- control keys, *See* accelerator keys
- convert entities 2-6, 2-14
  - silhouettes 2-29
- converting sketch entities to construction geometry 3-8
- coordinate system, creating 3-7
- copy and paste entire sketch 2-30
- copy feature 1-22, 5-7
- copy options wizard 1-8
- coradial relation 2-39
- cosmetic thread 10-25
- creating
  - assemblies 6-2
  - assembly features 7-16
  - cavities 7-18
  - components within assemblies 7-6
  - drawings 9-3
  - DXF/DWG mapping file 14-14
  - exploded view of assembly 6-35
  - extrusions 4-7
  - mating relationship 6-20
  - molds 7-18
  - new sketch 2-2
  - parts within assemblies 7-6
- crosshatch
  - automatic, assembly section views 9-26
  - faces and areas 10-40
  - in section view 9-23
- crosshatch options A-3
- cursor
  - edge selection 1-24
  - face selection 1-24
  - inferencing 2-27
- curvature 5-31, A-3
  - options A-19
- curve, split 2-20
- curves 3-9
  - from a file 3-16
  - helix 3-11
  - IGES options 14-8
  - projected 3-9, 3-10
  - silhouette split line 3-14
  - split line 3-12
- custom file properties
  - from model in a drawing view 9-4
  - in a design table 8-16
  - in bill of materials 10-36
  - in drawing templates 9-7
- custom properties 5-21
- custom symbol 10-41
- customize
  - keyboard shortcut keys 1-17
  - sketch grid 2-27
  - SolidWorks 1-7
  - toolbars 1-16
- customizing
  - assemblies 6-49
  - drawing template 9-5
  - grid A-16
  - headers and footers 1-34
- cut
  - by thickening a surface 4-57
  - end conditions 4-3–4-5
  - extrude 4-3
  - loft 4-15
  - revolve 4-9
  - sweep 4-10
  - thin feature 4-8
  - with surface 4-57
- cut and paste
  - drawing views 9-32
  - sketch entities 2-30

## D

- dangling
  - dimension or relation 1-20, 2-45
  - relations, reattaching 2-42
  - sketch geometry 2-44
- datum feature symbol 10-27
- datum tag, *See* datum feature symbol
- datum target symbol 10-28
- decimal separator A-6

## Index

- definition, edit 5-2
- deformed surface
  - creating 4-37
  - editing 4-39
  - shape 4-37
- deleting
  - assembly components 6-8
  - background image 14-18
  - configurations 8-5
  - design table 8-12
  - feature in multi-feature pattern 4-49
  - mating relationship 6-26
  - pattern instance 4-49
- density 5-25
- density options A-18
- dependencies
  - finding 6-10
- dependency editing 5-13
- derive component part 7-20
- derived part
  - base part 5-16
  - from assembly component 5-17
  - from palette part 11-14
  - mirror part 5-16
- derived sketch 2-32
- design methodologies for assemblies 6-2
- design table 8-8
  - creating in Excel 8-9
  - deleting 8-12
  - displaying in a drawing 8-18
  - editing 8-11
  - inserting a new table 8-10
  - specifying parameters 8-12
  - summary of parameters 8-17
- detail view 9-18
  - modifying 9-19
- detailing
  - options 10-2, A-4
  - overview 9-1
- dimensions
  - access control 5-8
  - adding 2-33
  - align collinear/radial 10-9
  - align ordinate 10-8
  - align parallel/concentric 10-9
  - angular 2-34
  - baseline 10-6
  - between arcs and circles 2-36
  - center text 10-10
  - changing 2-33, 2-35
  - copy 10-8
  - dangling 2-42
  - decimal separator A-6
  - display options A-4
  - drive with equations 2-35
  - driven and driving states A-14
  - font A-5
  - group 10-9
  - hide or show in drawings 10-17
  - hide or show in parts 5-23
  - import from model 10-4
  - in design tables 8-13
  - jog ordinate 10-7
  - modify 1-31, 2-35
  - modify text 10-16
  - modify value 10-12
  - modifying witness lines 10-11
  - move 10-8
  - offset text 10-11
  - options A-5
  - override on drag 2-21
  - parallel (in drawing) 10-5
  - point-to-point 2-34
  - preview during creation 2-34
  - properties 2-36, 10-12
  - reference 10-5
  - spin box 1-31, 2-35
  - standards A-4
  - to a silhouette 2-29
  - to midpoints 10-6
  - to silhouette edges 10-6
  - tolerance 10-16
  - true length of arc 2-36
  - ungroup 10-10
  - visibility 10-17
  - witness line control 10-13
- direct edit of sketch 2-45
- display
  - annotations in parts 5-22
  - arc centerpoints A-14
  - axis 3-6
  - background image 14-18
  - curvature 5-31
  - mode in drawing views 9-33
  - planes 3-5
  - quality A-18

- reference geometry 3-16
- section view of model 5-23
- status bar 1-31
- tangent edges in drawings 9-34
- temporary axes 3-6
- toolbars 1-16
- display quality options A-18
- display/delete relations 2-41
- dissolving a library feature 11-5
- distance mating 6-19
- dock toolbars 1-17
- document
  - icon 1-15
  - open from another format 14-2
  - windows 1-14
- document controls A-15
- dome 4-36
- draft 3-12, 4-7, 4-8, 4-29
  - creating angles 4-29
  - neutral plane 4-29
  - parting line 4-30
    - step draft 4-31
  - while extruding 4-5
- dragging
  - from Internet Explorer 1-11
  - parts into assemblies 6-6
  - selecting items 1-24
  - with feature handles 5-5
- dragging and dropping
  - features 1-22
- drawing
  - balloon callout 10-38
  - bill of materials 10-31
  - break lines 9-20
  - cosmetic threads 10-25
  - creating 9-3
  - datum feature symbol 10-27
  - datum target symbol 10-28
  - dimensions, moving 10-8
  - displaying a design table 8-18
  - displaying model dimensions 10-4, 10-18
  - displaying reference geometry 10-17
  - hole callout 10-24
  - import model dimensions 10-4
  - line format 9-36
  - options A-8
  - overview 9-1
  - ownership of drawing items 9-5
  - printing 9-37
  - rulers 9-9
  - selected view 9-10
  - sheet metal part 12-23
  - surface finish symbol 10-26
  - tangent edge display 9-34
  - updating 9-29
  - weld symbols 10-39
  - window 1-13, 9-9
- drawing sheet
  - multiple 9-4
  - properties 9-3
- drawing template
  - customizing 9-5
  - edit 9-5
  - save 9-7
- drawing view
  - activating 9-10
  - align edge 9-31
  - aligned section 9-22
  - aligning 9-31
  - auxiliary 9-15
  - broken 9-20
  - cut, copy, and paste 9-32
  - detail 9-18
  - display options A-8
  - dynamic activation A-9
  - empty 9-39
  - hide/show 9-33
  - moving 9-30
  - named 9-16
  - projection 9-14
  - properties 9-28
  - relative to model 9-17
  - rotating 9-31
  - section 9-21
  - standard 3 view 9-12
  - visibility 9-32
- driven dimension 2-45
- duplicate entities 14-9
- DXF/DWG files 14-13
  - attach dimensions 14-13
  - export 14-6
  - export options 14-14
  - import 14-2, 14-13
  - mapping 14-14
    - sketch from drawing 14-13
- dynamic drawing view activation 9-10
- dynamic highlighting 1-23

## Index

### E

- edges options A-10
- edit
  - auto reliefs 12-22
  - components within assemblies 7-7
  - definition of feature 5-2
  - design table 8-11
  - drawing template 9-5
  - exploded view 6-37
  - externally referenced part 5-17
  - hole 4-32
  - joined part 7-25
  - mating relationship 6-22
  - referenced file locations 6-14
  - seed feature of pattern 4-48
  - sketch 2-31
  - sketch plane 2-32
  - sub-assembly in context 6-33
- edit configuration 8-5
- edit imported document 14-4
- elbow parts, for routing 13-8
- ellipse 2-5, 2-9
  - centerpoint 2-5
- embedding an OLE object 1-35
- empty drawing view 9-39
- end condition dialog box 4-3, 4-3-4-5
- end types 4-3
- entity check 5-19
- entity types
  - IGES 14-7, 14-8
- envelope
  - selection criteria 6-46
  - show/hide 6-46
- equal relation 2-39
- equations 2-35, 5-9
  - mathematical functions 5-10
  - trigonometric functions 5-11
- errors in solving mates 6-27
- Excel file, saving a BOM 10-36
- exit
  - sketch 2-32, 3-5
- exploded view
  - autoexplode 6-35
  - collapsing 6-37
  - creating 6-35
  - editing 6-37
  - step editing tools 6-35
- exploding an assembly 6-35

- export options 14-6
  - ACIS 14-11
  - DXF/DWG 14-14
  - IGES 14-8
  - Parasolid 14-11
  - STL 14-12
  - TIFF 14-17
  - VRML 14-17
- extend sketch element 2-17
- external references 1-19, 7-2
  - derived parts 5-16, 5-18
  - editing referenced document 5-17
  - folder list 7-4
  - listing 7-2
  - options 7-3, A-11
  - search order A-12
  - searching for 7-4
  - update path 7-2
- extrude 4-2
  - base 4-3
  - boss 4-3
  - cut 4-3
  - end conditions 4-3-4-5
  - solid feature 4-7
  - surface 4-7, 4-51
  - thin feature 4-7
- extruded text
  - changing location 2-13
  - editing 2-14
  - on a part 2-13

### F

- fabricated piping, *See* routing
- face blend fillet 4-25
- face properties 5-4
- family of models 8-8
- feature 4-1
  - assembly feature 7-16
  - chamfer 4-28
  - circular pattern 4-42
  - copy 1-22
  - copying 1-21, 5-7
  - created from a surface 4-56
  - creating within assembly 7-16
  - dome 4-36
  - draft 4-29
  - drag and drop 1-21
  - edit definition 5-2

- extrude 4-2, 4-7
- feature palette 11-6
- fillet 4-23
- handles 5-5
- hole 4-32
- in-context 7-2
- legal sketch contours 2-25
- library feature 11-2
- linear pattern 4-41
- loft 4-15
- mirror 4-44
- move 1-22
- moving 5-5
- naming 1-18
- out-of-context 7-2
- pattern
  - circular 4-42
  - geometry 4-46
  - linear 4-41
  - mirror 4-44
  - mirror all 4-45
  - mirror pattern 4-44
  - pattern of patterns 4-45
- properties 5-4
- renaming 1-18
- reordering 1-21
- revolve 4-9
- rib 4-35
- scale 4-33
- scope 7-16
- shape 4-37
- shell 4-34
- suppressing 5-13
- suppressing, in a design table 8-14
- sweep 4-10
- thin 4-6, 4-7
- time-dependent 7-9
- unsuppressing 5-13
- updating in context 7-2
- feature palette
  - adding a feature to a part 11-12
  - adding a part to an assembly 11-13
  - applying a sheet metal forming tool 11-14
  - creating a derived part 11-14
  - creating items 11-9
  - displaying 11-7
  - editing items 11-11
  - navigating 11-8
  - organizing palette items 11-8
  - path for routing parts 13-12
  - requirements for palette features 11-6
  - search paths A-12
- feature recognition, FeatureWorks 14-20
- feature-based mating 6-24
- FeatureManager design tree 1-12, 1-18, 1-21, 1-25
  - arrow key navigation 1-20, A-13
  - component names 6-13
  - conventions 1-19
  - display by dependencies 6-5
  - display by features 6-5
  - displaying mates with components 6-5
  - external references 1-19
  - go to feature (in tree) 1-18, 1-22
  - in assemblies 6-3
  - options 1-20
  - symbols 1-19
  - text search 1-18
- FeatureWorks 14-19
  - automatic feature recognition 14-22
  - interactive feature recognition 14-21
- file
  - custom properties 5-21
  - derive component part 5-17
  - open from another format 14-2
- fillet 4-23
  - automatic in 3D sketch 13-12
  - constant radius 4-23
  - face blend 4-25
  - in a sketch 2-6, 2-15
  - overflow type 4-27
  - variable radius 4-24
- fillet/round 4-23
- find dependencies 6-10
- first angle projection 9-4
- fittings
  - adding to a route 13-17
  - alignment 13-17
  - custom elbows 13-12
  - elbows 13-8
  - flanges 13-10
  - multi-port 13-11
  - reducers 13-10
  - required features 13-3
- fix relation 2-39
- fixing
  - components 6-17
  - parts 6-17

## Index

flange parts, for routing 13-10

floating

components 6-17

parts 6-17

folders list A-12

follow path 4-11

font

dimensions A-5

footers and headers, customizing 1-34

force rebuild 1-32

form feature 12-27

format, lines in drawings 9-36

forming tools

applying 12-27

applying from feature palette 11-14

controlling access to dimensions 12-27

creating 12-24

editing 12-24

Feature Palette 12-27

indicating opening 12-26

minimum radius of curvature 12-25

positioning sketch 12-26

samples 12-24

stopping surface 12-25

fully defined sketch 2-44

functions in equations 5-10

## G

general options A-13

geometric relations 2-37

adding 2-37, 2-38

automatic creation 2-37

display/delete 2-41

removing 2-41

geometry-based mates 6-23

go to feature (in tree) 1-18

grid

align 2-21

options A-16

snap 2-27, A-16

guide curves

used in a loft 4-18, 4-19

used in a sweep 4-11

## H

handles, feature 5-5

headers and footers, customizing 1-34

helix 3-11

help point 4-25

help, online 1-8

hems 12-18

hidden in gray (HLG) 1-27

hidden items, selecting 1-25

hidden lines removed (HLR) 1-27

hide

axes 3-6

background image 14-18

component 6-39

dimensions 5-23

drawing view 9-33

edges in drawing views 9-33

feature 1-22

planes 3-5

status bar 1-31

toolbars 1-16

highlighting 1-23

hole

feature 4-32

simple 4-3, 4-32

wizard 4-3, 4-32, 4-33

hole callout 10-24

hollowing. *See* shell

horizontal

relation 2-39

horizontal break lines 9-20

hyperlink in a note 10-22

hyperlink, open file from Internet 1-11

## I

iconize documents 1-15

IGES files

2D geometry 14-10

3D curves 14-8

b-splines 14-8

edit imported document 14-4

entity types 14-7, 14-8

export 14-6

export options 14-8

import 14-2, 14-9

import options 14-3

importing a surface 14-2

levels 14-9

parametric splines 14-8

trimmed surfaces 14-8

- import 14-2
  - model items (annotations) 10-18
  - model items (dimensions) 10-4
  - options 14-3
  - reference geometry 10-17
  - sketch, constrain all 2-4, 2-43
  - surface from file 14-2
- in-context features
  - saving 7-5
  - updating 7-2
- increment value, spin box A-17
- inertia tensor 5-25
- infer from model A-14
- inferencing
  - cursors 2-27
  - lines 2-28
  - to the assembly origin 6-8
- inferencing lines
  - automatic 2-29
- insert
  - sketch from drawing 9-39
- inserting
  - base part 5-16
  - components, into assemblies 6-6
  - mirror part 5-16
  - new design table 8-10
  - object 1-35
  - parts, into assemblies 6-6
  - picture from TIFF file 14-18
- inspection dimension 10-15
- installation
  - MCD (Mini-Client Driver) enable 1-3
  - procedure 1-2
  - required information 1-2
  - requirements 1-2
- interactive feature recognition,
  - FeatureWorks 14-21
- interference detection, in assemblies 6-38
- Internet, open hyperlinked file 1-11
- intersection
  - relation 2-39
- in-use configuration 8-7
- invalid geometry 2-44
- invalid solution found 2-44

## J

- joining parts 7-24

## K

- keep normal constant 4-11
- keyboard shortcuts 1-32
  - customizing keys 1-17
- k-factor 12-7
- knit surfaces 4-54
  - seed face option 4-54
  - thicken 4-54

## L

- language options A-15
- layers, IGES. *See* levels, IGES
- layout sketch in assembly 7-14
- length unit of measure A-16
- levels, IGES 14-9
- library features
  - adding color 11-5
  - adding to part 11-3
  - dissolving 11-5
  - editing 11-4
- lighting 5-27
  - adding light sources 5-27
  - light properties 5-28
  - light types 5-27
- lighting conditions, advanced A-3
- lightweight
  - advantages 6-41
  - components in assemblies 6-40
- line 2-5, 2-7
  - and point plane 3-3
  - weight setup 1-33
- line font
  - in assembly drawings 9-35
  - options A-17
- line format toolbar 9-35
- linear pattern 4-41
  - spacing 4-41
  - total instances 4-41
- linear step and repeat 2-18, 2-19
- link to property 9-6, 9-8, 10-21
- linking an OLE object 1-35
- list external references 7-2
- list named entities 6-17
- load referenced documents 5-18
- lock sheet focus 9-10
- lock view focus 9-10
- loft
  - advanced smoothing 4-16, 4-19
  - base 4-15



## Index

- boss 4-15
  - center line 4-20
  - close along loft direction 4-16
  - cut 4-15
  - maintain tangency 4-16
  - profiles
    - non-planar 4-16
    - planar 4-15
  - simple 4-15
  - surface 4-15, 4-52
  - tangency options 4-21
  - with guide curve 4-18, 4-19
  - with split line 4-17
- loops, selecting 1-24
- ## M
- macros
- API code shortcut B-4
  - playing B-2
  - recording B-2
- mailing
- assembly documents 6-52
  - drawing documents 9-38
  - part documents 5-32
- maintain tangency
- with lofts 4-16
  - with sweeps 4-12
- mapping, DXF/DWG 14-14
- mass properties 5-25
- mate reference 6-25
- material properties options A-18
- material properties, advanced 5-3
- mathematical functions 5-10
- mathematical relations 2-35
- mating
- adding 6-20
  - alignment conditions 6-21
  - angle 6-19
  - automatic 6-22
  - coincident 6-19
  - components in assemblies 6-19
  - concentric 6-19
  - creating 6-20
  - deleting 6-26
  - displaying mates with components 6-5
  - distance 6-19
  - editing relationships 6-22
  - errors 6-27
  - feature-based 6-24
  - generate entity names 6-16
  - geometry-based 6-23
  - mate reference 6-25
  - mategroups 6-19
  - modifying 6-22
  - offset 6-19
  - parallel 6-19
  - parts in assemblies 6-19
  - pattern-based 6-25
  - perpendicular 6-19
  - preserve mates when replacing
    - components 6-16
  - relationships 6-19
    - alignment 6-19
    - assembly constraints 6-19
  - showing mates for each component 6-5
  - smart 6-22
  - tangent 6-19
  - types 6-19
- measure 5-11
- changing units 5-12
  - options 5-12
- measuring in assemblies 6-18
- merge points relation 2-40
- merging component bodies 7-24
- mid-plane extrusion 4-4
- midpoint
- relation 2-39
- midpoint selection 1-24
- minimum radius of curvature 12-25
- mirror 2-6, 2-14
- all 4-45
  - derive mirrored part 5-16
  - feature 4-44
  - part 5-16
  - part around a planar face 4-45
  - pattern 4-44
  - sketch elements 2-6, 2-14
- model
- cosmetic threads 10-25
  - modify sketch 2-22
  - modify text of dimensions 10-16
  - modify value of dimensions 10-12
  - modify mating relationships 6-22
- molds 7-18
- complex parting line 7-21
  - core and cavity 7-22

- creating a cavity 7-19
  - parting plane 7-20
  - scaling factor 7-18
- move
  - sketch 2-22
  - sketch entities 2-30
- move view (pan) 1-27
- move/size feature tool 5-5
- moving
  - components in assemblies 6-18
  - dimensions 10-8
  - drawing views 9-30
  - feature 1-22
  - toolbars 1-17
  - with feature handles 5-5
- multiple drawing sheets 9-4
- multiple items, selecting 1-24
- multiple views 1-30
- multi-threaded retrieval 1-10

## N

- named entities, list 6-17
- named view in a drawing 9-16
- named view orientation 1-29
- named views
  - exploded view 6-37
- naming features 1-18
- neutral plane draft 4-29
- new
  - drawing document 9-3
  - window 1-30
- no solution found 2-44
- not solved 2-44
- note
  - hyperlink 10-22
  - link to document properties 10-21
  - multiple leaders 10-23
  - options A-5
  - parametric 10-20

## O

- objects
  - embedding 1-35
  - inserting 1-35
  - linking 1-35
  - OLE 1-35

- offset
  - from surface extrusion 4-3
  - plane 3-2
  - surface 4-53
- offset entities 2-6, 2-16
  - silhouettes 2-29
- OLE automation interface B-1
- OLE object
  - display contents 1-37
  - drag and drop 1-37
  - drawing sheet selection 1-37
  - embedding 1-35
  - inserting 1-37
  - linking 1-35
- on surface plane 3-3
- online help 1-8
- open
  - document from another format 14-2
  - DXF/DWG mapping file 14-15
- opening
  - assemblies 6-45, 6-49
  - parts within assemblies 7-8
  - sketch on a plane 3-2
  - SolidWorks documents 1-9
    - Quick View mode 1-10
- opening SolidWorks documents
  - multi-threaded retrieval 1-10
  - view-only 1-10
- options
  - AutoRecover 1-8
  - color A-2
  - copy registry information 1-8
  - crosshatch A-3
  - detailing 10-2, A-4
  - document controls A-15
  - drawings A-8
  - edges A-10
  - export 14-6
    - ACIS 14-11
    - DXF/DWG 14-14
    - IGES 14-8
    - Parasolid 14-11
    - STL 14-12
    - TIFF 14-17
    - VRML 14-17
  - external references 7-3, A-11
  - FeatureManager design tree A-13
  - FeatureWorks 14-20

## Index

- general A-13
    - FeatureManager design tree 1-20
  - grid A-16
  - grid/units 2-27
  - import 14-3
  - language A-15
  - line font A-17
  - material properties A-18
  - measurement 5-12
  - model A-13
  - performance A-18
  - reference geometry A-20
  - routing 13-12, A-21
  - sketch A-14
  - view A-13
  - virtual sharp 2-12
  - ordinate dimensions 10-7
  - orientation, view 1-29
  - orientation/twist control 4-11, 4-13
  - origin
    - inference in assembly 6-8
    - sketch 2-27
  - out-of-context features 7-2
  - over defined 2-44
    - sketch 2-44
  - over defining A-14
    - dimension or relation 2-45
  - overflow types, fillet feature 4-27
  - overlapping parts in assemblies 6-38
  - override dimensions on drag 2-21, A-14
- P**
- page
    - orientation 1-33
    - setup 1-33
  - palette, feature 11-6
  - pan 1-27
  - paper margins, setup 1-33
  - parabola 2-10
  - parallel
    - mating relationship 6-19
    - relation 2-39
  - parallel dimensions
    - align 10-9
  - parallelogram 2-5, 2-8
  - parametric note text 10-20
  - Parasolid files 14-10
    - export 14-6
    - export options 14-11
    - import 14-2
  - parent/child relationships 5-5
  - parentheses, show on reference
    - dimensions 10-3
  - part number, in a design table 8-15
  - partial section view 9-25
  - parting line 3-12
  - parting line draft 4-30
  - parting plane, in molds 7-20
  - parts
    - adding features from feature palette 11-12
    - adding to assemblies 6-6
    - configurations in an assembly 8-7
    - creating configurations 8-2
    - creating within assemblies 7-6
    - derive component part 5-17, 7-20
    - derive mirrored part 5-16
    - derived from feature palette 11-14
    - displaying annotations 5-22
    - dragging into assemblies 6-6
    - editing configurations 8-5
    - editing within assemblies 7-7
    - fixing 6-17
    - floating 6-17
    - insert base part 5-16
    - inserting into assemblies 6-6
    - joining 7-24
    - lightweight 6-40
    - list, *See* bill of materials
    - mating in assemblies 6-19
    - opening within assemblies 7-8
    - overlapping in assembly 6-38
    - positioning in assemblies 6-17
    - replacing in assemblies 6-12
    - resolved 6-40
    - suppressing a feature 5-13
    - suppressing in assemblies 6-40
    - verifying in assembly 6-10
  - paths, of assembly components 6-10
  - pattern
    - circular 4-42
    - circular step and repeat 2-19
    - color 4-46
    - component pattern in assembly 6-8, 7-11
    - deleting feature from multi-feature 4-49

- deleting instance 4-49
- editing seed feature 4-48
- geometry 4-46
- linear 4-41
- linear step and repeat 2-18, 2-19
- mirror all 4-45
- mirror feature 4-44
- mirror pattern 4-44
- of patterns 4-45
- restoring deleted instance 4-49
- vary sketch 4-47
- pattern-based mates 6-25
- penetration of pipes 13-18
- performance options A-18
- perpendicular
  - curve at point plane 3-3
  - mating 6-19
  - relation 2-39
- perspective
  - view 1-27
- PhotoWorks 1-38
- picture, inserting a TIFF image 14-18
- pierce relation 2-40
- pipe parts, for routing 13-7
- pipng, *See* routing
- pitch of helix 3-11
- planar surfaces, bounded 4-50
- plane and point plane 3-3
- planes
  - at angle 3-2
  - construction 3-2
  - default names A-20
  - display 3-5
  - naming 3-5
  - parting, in molds 7-20
  - preferences 3-5
  - renaming 3-5
  - setting up 3-2
  - tangent to surface 3-3
  - types 3-2
  - visibility 3-5
- plotter setup 1-33
- point
  - connection 13-3
  - reference 2-5, 2-12
  - route 13-3
  - virtual intersection 2-12
- point light 5-27
- pointer
  - inferencing 2-27
- point-to-point dimensions 2-34
- positioning
  - components in assemblies 6-17
  - parts in assemblies 6-17
- positioning sketch 12-26
- preferences
  - general
    - FeatureManager design tree 1-20
- preparation for printing 1-33
- preview of dimensions 2-34
- principal axes 5-25
- print
  - drawings 9-37
  - options, setup 1-33
- printer setup 1-33
  - color 1-34
- printing preparation 1-33
- program window 1-12
- projected curve 3-9, 3-10
  - from orthogonal sketches 3-10
  - project sketch on model 3-9
  - replacing 3-10
- projection view 9-14
- projection, first or third angle 9-4
- properties
  - bill of materials 10-33
  - common 3-8
  - custom 5-21
  - custom, in a design table 8-16
  - dimension 2-36, 10-12
  - drawing view 9-28
  - face 5-4
  - feature 5-4
  - lighting 5-28
  - mass 5-25
  - section 5-26
  - section line 9-24
  - tolerance A-5
- PropertyManager 2-45

**R**

- radiate surface 4-53
- radiated surface
  - used to cut a mold 7-21
- radius of curvature 5-31
- rapid prototyping files. *See* STL files
- reattaching dangling dimensions 2-42

## Index

- rebuild 1-32
  - with verification 5-19
- rebuild icon 1-21
- recognizing features, FeatureWorks 14-20
- rectangle 2-5, 2-7
- rectangle at an angle 2-8
- reducer parts, for routing 13-10
- reference dimensions 10-5
  - show parentheses 10-3
- reference geometry
  - curves 3-9
  - display 3-16
  - in drawings 10-17
  - options A-20
- reference, external 5-16, 7-2
- reference, mate 6-25
- referenced configuration 8-7
- referenced file locations in assemblies 6-14
- references, finding 7-4
- regenerate model 1-32
- regenerate symbol 1-21
- registry copy tool 1-8
- relations
  - automatic 2-37
  - coincident 2-39
  - collinear 2-39
  - concentric 2-39
  - coradial 2-39
  - display/delete 2-41
  - equal 2-4, 2-39, 2-43
  - fix 2-39
  - geometric 2-37
  - horizontal 2-39
  - intersection 2-39
  - mathematical 2-35
  - merge points 2-40
  - midpoint 2-39
  - parallel 2-39
  - perpendicular 2-39
  - pierce 2-40
  - symmetric 2-39
  - tangent 2-39
  - vertical 2-39
- relationship codes, in assemblies 6-4
- relationships
  - parent/child 5-5
- relative to model view 9-17
- reload component 6-13
- removing assembly components 6-8
- renaming features 1-18
- reordering assembly items 6-32, 7-13
- reordering features 1-21
- reorganize components 6-32
- replace assembly component 6-12
  - edit component properties 6-13
  - open with new references 6-14
  - preserve mates 6-16
  - reload/replace 6-13
  - save with new references 6-15
- replacing, background image 14-18
- requirements for installation 1-2
- resizing a feature 5-5
- resolve result ambiguity 7-23
- resolved
  - components in assemblies 6-40
- restoring pattern instance 4-49
- retrieval, multi-threaded 1-10
- reusing parts, in assemblies 6-4
- revert to earlier state 5-15
- revolve
  - base 4-9
  - boss 4-9
  - cut 4-9
  - feature 4-9
  - surface 4-9, 4-51
  - thin feature 4-9
- rib 4-35
  - with draft angles 4-35
- right-mouse menu 2-30
- rip 12-14
- roll back
  - in assemblies 7-13
- roll forward 5-15
- rollback 5-15
- rotate
  - component in assembly 6-18
  - drawing views 9-31
  - feature 5-5
  - sketch 2-22
  - view 1-27
- round model edge 4-23
- route point 13-3, 13-6
- routing 13-1
  - 3D sketch 13-3
  - adding fittings manually 13-17
  - automatic sketch fillets 13-12

- connection points 13-3
- creating a route 13-14
- drawing views of routes 13-20
- editing a route 13-16
- elbow parts 13-8
- fitting alignment 13-17
- flange parts 13-10
- general procedure 13-2
- library folder path 13-12
- multi-port fittings 13-11
- options 13-12, A-21
- overview 13-2
- part requirements 13-3
- pipe parts 13-7
- pipe to pipe penetration 13-18
- reducer parts 13-10
- reorganizing components 13-19
- route points 13-3

rulers, in drawings 9-9

## S

- sample projects, API B-4
- sat files. *See* ACIS files
- satisfied dimension 2-45
- save
  - assemblies 6-10
  - assembly with in-context features 7-5
  - assembly with new references 6-15
  - DXF/DWG mapping file 14-14
  - snapshots 1-31
  - template 9-7
- scale
  - model 4-33
  - sketch 2-22
- scaling factor, for mold cavity 7-18
- scan equal 2-4, 2-43
- scientific notation 5-12, 5-25
- scope of assembly feature 7-16
- scope of assembly section view 9-26
- section arrow display, alternate A-4
- section line properties 9-24
- section properties 5-26
- section view
  - aligned, in drawing 9-22
  - auto hatching 9-26
  - centerline display 9-24
  - crosshatch 9-23
  - in drawing 9-21
  - modifying 9-23
  - of a part 5-23
  - of an assembly 9-26
  - scope 9-26
  - troubleshooting 9-27
- seed surface 4-55
- select other 1-25
- select tool 2-3
- selected view 9-10
- selecting
  - models for drawing views 9-12
- selecting items 1-23–1-25
  - advanced component selection 6-47
  - coincident 1-25
  - consecutive items in list 1-25
  - hidden 1-25
  - loops 1-24
  - midpoint 1-24
  - multiple 1-24, 1-25
- selection criteria 6-47
- selection filter 1-23
  - hot keys 1-23
  - toolbar 1-23
- sending
  - assembly documents 6-52
  - drawing documents 9-38
  - part documents 5-32
- service packs 1-5
- setting up construction planes 3-2
- shaded view 1-27
- shape
  - bend 4-38
  - boundary tangents 4-37
  - constraining 4-38
  - creating 4-37
  - curve influence 4-38
  - deflating 4-38
  - deformed surface 4-37
  - editing 4-39
  - inflating 4-38
  - resolution 4-39
  - stretch 4-38
- sheet metal
  - adding features 12-17
  - applying forming tools 11-14
  - auto relief 12-9
  - bend across multiple tabs 12-19
  - bend allowance 12-7
  - bend table 12-8
  - bend types 12-3

## Index

- creating part
    - using a flat model 12-16
    - using cylindrical or conical faces 12-13
    - using filleted corners 12-12
    - using sharp bends 12-10
  - cut across bend 12-17
  - design tips 12-24
  - designing parts 12-2
  - drawings 12-23
  - editing
    - auto reliefs 12-22
    - bends 12-20
  - features in FeatureManager design tree 12-4
  - fixed edge 12-13
  - fixed face 12-11
  - flat bends 12-3
  - flat pattern 12-22
  - forming tools 12-24
  - hems 12-18
  - k-factor 12-7
  - link to thickness 12-10
  - overview 12-1
  - recommended design approach 12-3
  - rip 12-14
  - rolling back and rebuilding 12-6
  - round bends 12-3
  - sharp bends 12-3
  - tab 12-17
- sheet setup, drawing 9-3
  - sheet, drawing 9-3
  - shell 4-34
  - shortcut menu 1-22
  - shortcuts, keyboard 1-32
  - show
    - assembly hierarchy 6-5
    - axes 3-6
    - component 6-39
    - dimensions 5-23
    - drawing view 9-33
    - feature 1-22
    - planes 3-5
  - show parentheses 10-3
  - show/hide components, advanced 6-48
  - significant digits 14-4
  - silhouette
    - dimensioning to 2-29
    - edges 2-29
  - simple hole 4-32
  - simplified
    - assemblies 8-6
    - parts, opening in assembly 8-6
  - simplified thread, *See* cosmetic thread
  - simplify spline 2-11
  - simplifying complex assemblies 6-39
  - sketch
    - 2D, in drawings 9-39
    - check for feature usage 2-24
    - complexity of 2-49
    - copy and paste 2-30
    - cut and paste entities 2-30
    - derived 2-32
    - edit 2-31
    - edit plane 2-32
    - entity tools 2-7
    - exit 2-32, 3-5
    - fillet 2-15
    - for assembly layout 7-14
    - grid 2-27, A-16
    - inferencing lines 2-28
    - inferencing pointers 2-27
    - legal contour types 2-25
    - mode 2-26
    - modify 2-22
    - on a part 2-31
    - on a plane 3-2
    - options A-14
    - plane, translucent 2-31
    - relations 2-37
    - scale 2-22
    - starting 2-2, 2-48
    - symmetric 2-6, 2-14
    - tools, accessing 2-6
    - underive 2-32
    - window 2-2, 2-26
  - sketch entities
    - direct edit 2-45
    - properties 2-45
    - PropertyManager 2-45
    - split curve 2-20
  - sketch entity
    - arcs 2-9
    - centerline 2-12
    - centerpoint ellipse 2-10
    - circle 2-8
    - circular sketch and repeat 2-19
    - ellipse 2-9

- line 2-7
- linear sketch and repeat 2-18, 2-19
- parabola 2-10
- parallelogram 2-8
- point 2-12
- properties 2-46
- rectangle 2-7
- rectangle at an angle 2-8
- spline 2-10
  - tools, accessing 2-5
- sketch from drawing 14-13
- sketched curve
  - from model edge 2-6, 2-14
- sketching
  - 3D 2-50
- sla files. *See* STL files
- slant witness lines 10-11
- smart mates 6-22
- snap behavior A-16
- solid feature 4-7
- SolidWorks
  - installation 1-2
  - terminology 1-12–1-13
  - web site 1-5
  - windows 1-12–1-14
- spin box
  - calculator 1-31
  - dimensions 1-31, 2-35
  - increment value 1-31, 2-35, A-17
- spline 2-10
  - points 2-11
  - reshape 2-11
  - simplify 2-11
- split curve 2-20
- split line
  - loft 4-17
  - planar 3-12
  - silhouette 3-14
- split windows 1-30
- spot light 5-27
- standard 3 view 9-12
- standard view toolbar 1-28
- starting a sketch 2-2, 2-48
- status bar 1-31, 2-26
  - display or hide 1-31
- step draft 4-31
- STEP files 14-11
  - edit imported document 14-4
  - export 14-6
  - import 14-2
  - import options 14-3
- Stereolithography
  - files, how to create 14-12
- STL files 14-12
  - export 14-6
  - export options 14-12, A-18
- sub-assemblies
  - routing 13-2
- sub-assembly 6-28
  - adding components to 6-28
  - changing component order 6-33
  - creating 6-28
  - dissolving 6-31
  - editing in context 6-33
  - form new sub-assembly 6-29
  - inserting a new, empty sub-assembly 6-28
  - limitations when modifying 6-30
  - reorganizing hierarchy 6-32
- suppressing
  - all components 6-45
  - components in assemblies 6-40
  - components, in design table 8-14
  - features 5-13
  - features in a design table 8-14
- surface
  - radiate 4-53
- surface area 5-25
- surface errors 14-4
- surface finish symbol 10-26
- surface flaws 14-4
- surfaces
  - creating 4-50
  - cut by thickening surface 4-57
  - cut with surface 4-57
  - extrude 4-7, 4-51
  - IGES files 14-9
  - knit 4-54
  - loft 4-15, 4-52
  - loft to fill volume 4-55
  - offset 4-53
  - planar, with a boundary 4-50
  - revolve 4-9, 4-51
  - sweep 4-10, 4-52
  - thicken 4-54
  - thickening 4-56
  - using to create features 4-56



## Index

- sweep
    - advanced smoothing 4-12
    - base 4-10
    - boss 4-10
    - cut 4-10
    - feature 4-10
    - follow path 4-11
    - keep normal constant 4-11
    - maintain tangency 4-12
    - orientation/twist control 4-13
    - path 4-10
    - propagate along tangent edges 4-11
    - section 4-10
    - show intermediate profiles 4-13
    - surface 4-10, 4-52
    - with guide curves 4-11
  - symbols
    - custom 10-41
    - datum feature symbol 10-27
    - datum target symbol 10-28
    - surface finish 10-26
    - weld 10-39
  - symmetric relation 2-39
  - symmetrical part 2-6, 2-14, 4-45
  - system requirements 1-2
  - system separator A-6
- ## T
- table, design. *See* design table
  - tangent
    - mating relationship 6-19
  - tangent arc 2-5, 2-9
  - tangent edges in drawings 9-34
  - tangent relation 2-39
  - template
    - bill of materials 10-31
  - template, drawing 9-5
  - temporary axes 3-6
  - terms, used in SolidWorks 1-12
  - text on part 2-13
  - text search of FeatureManager design tree 1-18
  - thicken knit surfaces 4-54
  - thickening a surface 4-56, 4-57
  - thin feature 4-6, 4-7, 4-8
  - third angle projection 9-4
  - thread, cosmetic 10-25
  - three point arc 2-5
  - three point plane 3-2
  - through all extrusion 4-3
  - thumbnail graphics 1-10, A-15
  - TIFF files 14-17
    - export 14-6
    - export options 14-17
    - import 14-3, 14-18
  - tile windows 1-14
  - time-dependent feature 7-9
  - tips, time savers 1-31
  - tolerance
    - of dimensions 10-16
    - values A-5
  - toolbars 1-16
    - assembly 6-4
    - customizing 1-17
    - display or hide 1-16
    - icon size 1-17
    - line format 9-35
    - move 1-17
    - sketch 2-3
    - sketch entity tools 2-7
    - sketch relations 2-4
    - sketch tools 2-5, 2-6
    - standard view 1-28
    - view 1-26
  - tooltips 1-31
  - top-down design 6-2, 7-1
  - trailing zeros display A-4
  - translate sketch 2-22
  - translucent sketch plane 2-31
  - transparency quality A-19
  - trim sketch element 2-16
- ## U
- under defined 2-44
    - sketch 2-44
  - underive sketch 2-32
  - undo
    - view orientation 1-29
  - units of measure, default A-16
  - unlock sheet focus 9-10
  - unlock view focus 9-10
  - unsuppressing
    - a feature 5-13
  - up to next extrusion 4-3, 4-4
  - up to surface extrusion 4-3
  - up to vertex extrusion 4-4
  - update path 7-2

- updates, SolidWorks service packs 1-5
- updating drawings 9-29
- use named configuration 8-7
- user notes, in a design table 8-16
- user-defined symbols 10-41

## V

- variable
  - pattern 4-48
  - radius fillet 4-24
- vary sketch 4-47
- VDAFS files 14-11
  - edit imported document 14-4
  - export 14-6
  - import 14-2
  - import options 14-3
  - importing a surface 14-2
- verification on rebuild 5-19
- verifying, parts files in assembly 6-10
- vertical
  - relation 2-39
- vertical break lines 9-20
- view
  - align edge 9-31
  - aligned section 9-22
  - auxiliary 9-15
  - boundaries 9-11
  - broken 9-20
  - detail 9-18
  - display named view 1-29
  - drawing, hide/show 9-33
  - dynamic drawing view activation 9-10
  - empty 9-39
  - hidden lines in gray 1-27
  - hidden lines removed 1-27
  - move (pan) 1-27
  - multiple windows 1-30
  - named 9-16
  - options A-13
  - orientation 1-29
  - partial section 9-25
  - perspective 1-27
  - planes 3-5
  - projection 9-14
  - properties 9-28
  - relative to model 9-17
  - rotate 1-27
  - section 9-21
  - section of model 5-23

- selected 9-10
- shaded 1-27
- standard 3 view 9-12
- toolbar 1-26
- update 9-29
- wireframe 1-27
- zoom in/out 1-26
- zoom to area 1-26
- zoom to fit 1-26
- zoom to selection 1-26

- viewing
  - assembly by dependencies 6-5
  - assembly by features 6-5
  - assembly hierarchy 6-5
  - exploded view 6-37
  - unopened parts, assemblies, and drawings 1-10
- view-only mode 1-10
- virtual intersection point 2-12
- virtual sharp 2-12, A-6
- visibility
  - drawing views 9-32
- visibility of assembly components 6-39
- visibility, in design table 8-14
- volume 5-25
- VRML files 14-17
  - edit imported document 14-4
  - export 14-6
  - export options 14-17
  - import 14-2

## W

- weld
  - symbols 10-39
- welding
  - contact faces 7-27
  - general procedure 7-27
  - surface deltas 7-26
  - tips 7-28
  - types supported 7-26
  - weld symbols 7-28
- What's Wrong? functionality 1-20
- windows
  - arrange icons 1-15
  - cascade 1-14
  - close all 1-15
  - management 1-14
  - SolidWorks document 1-14

## Index

- tile horizontally 1-14
- tile vertically 1-14
- Windows95 zooming A-19
- wireframe 1-27
- with surface cut 4-57
- witness lines
  - change attachment point 10-11
  - control 10-13
  - slant 10-11
- World Wide Web 1-5

## X

x\_b, x\_t files. *See* Parasolid files

## Z

zoom

- in/out 1-26, 1-32
- to area 1-26
- to fit 1-26
- to selection 1-26