



SolidWorks® 2000

What's New

© 1995-2000, SolidWorks Corporation
300 Baker Avenue
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® and the SolidWorks logo are the registered trademarks of SolidWorks Corporation.

SolidWorks® 2000 is a product name of SolidWorks Corporation.

FeatureManager® is a jointly owned trademark of SolidWorks Corporation.

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 Solutions Co. Limited.

Other brand or product names are trademarks or registered trademarks of their respective holders.

Document Number: SWXWNENG021500

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.

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 © 1995-2000
D-Cubed Limited.

Portions of this software © 1992-2000
Summit Software Company.

Portions of this software © 1990-2000
LightWork Design Limited.

Portions of this software © 1995-2000
Spatial Technology Inc.

Portions of this software © 1998-2000
Geometric Software Solutions Co. Limited.

Portions of this software © 1999-2000
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



Introduction	ix
About This Book.	ix
Moving to SolidWorks 2000.	x
Chapter 1 SolidWorks Fundamentals	1-1
Tools and Toolbars	1-2
Document Templates	1-5
Opening a New Part.	1-5
Using Tabs.	1-6
FeatureManager Design Tree	1-7
Annotations	1-7
Duplicate Panel Display	1-7
PropertyManager	1-8
Shortcut Menu Functions	1-8
ConfigurationManager	1-8
Additional Shortcut Menu Options	1-8
Middle Mouse Button Functions	1-9
SolidWorks Options	1-9
Options Dialog Box Structure	1-9
Accessing the Options Dialog Box	1-10
New Options Structure	1-10
Changes in Options	1-11
SolidWorks Explorer.	1-11

Import and Export	1-12
Import Options	1-12
DXF/DWG Files	1-13
Parasolid Files	1-13
IGES Files	1-13
STEP Files	1-14
Display Performance	1-15
Chapter 2 Sketches	2-1
New Sketch Toolbar Tools	2-2
Text Tool	2-2
Construction Geometry	2-2
Polygon	2-2
Intersection Curve	2-4
Parametric and Editable Sketch Patterns	2-5
Circular Sketch Step and Repeat	2-5
Linear Sketch Step and Repeat	2-7
Splines	2-8
3D Sketching	2-11
Chapter 3 Surfaces	3-1
Surface Bodies	3-2
Extend Surfaces	3-3
Definitions for Extend Surfaces	3-3
Characteristics and Capabilities of Extend Surfaces	3-3
Extend Surfaces Samples	3-5
Trim Surface	3-7
Definitions for Surface Trim	3-7
Characteristics and Capabilities of Surface Trim	3-7
Trim and Extend Surfaces Samples	3-10
Fillet Surface	3-11
Surface Delete	3-13
Split Line on Surface	3-13
Knit Surfaces	3-13
Hide and Show Bodies	3-14

4-1

5-1

V

Balloons	6-10
Stacked Balloons.....	6-11
Bill of Materials	6-11
RapidDraft Drawings	6-12
Advantages of RapidDraft Drawings	6-12
View Updates	6-12
New Drawings and Existing Drawings.....	6-13
Operations Available in RapidDraft Drawings Without the Model	6-13
Operations Requiring the Model.....	6-14

Introduction

About This Book

There are many new features available in the SolidWorks® 2000 software. This book was created to highlight the new functionality and to help you learn these new features. This book is for experienced users and assumes that you have a good working knowledge of an earlier release of the SolidWorks software.

If you are new to using the SolidWorks software, you should contact your reseller for information about SolidWorks training classes.

Using This Book

Use this book in conjunction with the part, drawing, and assembly files provided.

- 1 Install the SolidWorks 2000 software.

Be sure to select the option for installing the **Example Files**. The example files for this book are placed in the *installation directory***samples\what's new** folder.

TIP: To install the *SolidWorks 2000 Getting Started* in .pdf format, select the option for installing the **Manuals** as well. The manuals are placed in the *installation directory***lang\your language\manuals** directory.

- 2 Create a new folder on your system, and copy the sample files into the local folder.

NOTE: Since some of the sample files are used with more than one example, *do not* save changes to these files.

- 3 Follow the instructions.

Go through this book from beginning to end, opening the proper part, drawing, or assembly document for each example, or use the Table of Contents or Index to locate topics of special interest to you.

Late Changes

We have tried to make this book as complete and accurate as possible. Refer to the *Read This First* and *Release Notes* that are shipped with your SolidWorks software for information that could not be included in this printed book. Also, refer to the **Overview of New Functionality in SolidWorks 2000** in the *Online User's Guide*.

Moving to SolidWorks 2000

Backup Copies

We recommend that you save backup copies of all SolidWorks documents (parts, assemblies, and drawings) before opening them in SolidWorks 2000. These documents are automatically converted to SolidWorks 2000 format when opened. Once converted and saved, the documents are not accessible in earlier releases of the SolidWorks software.

Converting Older SolidWorks Files to SolidWorks 2000

Because of changes to the SolidWorks files with the development of SolidWorks 2000, 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 returns 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 2000 format. Depending on how many files you have, the conversion process may take a while, but once it is done, the files will open more rapidly.

To access the Conversion Wizard, click the Microsoft **Start** button, select **Programs, SolidWorks 2000**, then **SolidWorks Utilities**. 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 sub-folder named "Solidworks Conversion Backup." The wizard asks you for the location of the files to be converted, and leads you through the 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.

Copy Options Wizard

After you customize your SolidWorks 2000 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 2000, SolidWorks Utilities, Copy Options Wizard**.
- 2 To capture your SolidWorks options information from the registry, click **An existing user** and select your name from the scroll list. Click **Next**.
- 3 Specify the destination for the options information:
 - **Users or Computers on the network.** As a system administrator on Microsoft® Windows NT®, you can copy the SolidWorks options directly to other selected users or computers on the network.
 - or –
 - **Create a Registry file.** You can create a **.reg** file that you can use with the standard Windows NT, Windows® 95, Windows 98, or Windows 2000 “regedit” utility to load the SolidWorks options on the computers of other users.

SolidWorks Service Packs

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 2000 software. To check for a new service pack, click **Help, Service Packs**, and click the **Check** button. Select the check box if you want the software to automatically check the SolidWorks Web site for a new service pack once a week.

SolidWorks 2000 Getting Started

The *SolidWorks 2000 Getting Started* book is on the SolidWorks CD-ROM in **.pdf** format. If you did not select the **Manuals** option when installing the software, you must reinstall the SolidWorks software to access the manuals.

To access the installed manuals, click **Help, SolidWorks 2000 Getting Started** when running the SolidWorks 2000 application.

SolidWorks Fundamentals

This chapter discusses some basic concepts used throughout the SolidWorks 2000 application. It provides an overview of the enhancements to the following topics:
















- ☐ Tools and toolbars
- ☐ Document templates
- ☐ FeatureManager design tree
- ☐ Shortcut menus
- ☐ Middle mouse button functions
- ☐ SolidWorks options
- ☐ SolidWorks Explorer
- ☐ Import and Export
- ☐ Display performance























Tools and Toolbars












There are six new toolbars: Align, Layer, Mold Tools, Surfaces, Curves, and Sheet Metal. New tools are distributed among the new, and the existing toolbars.

Some tools are not new, but were added to a new toolbar, in addition to remaining in their original location. For example, Draft and Scale, still found on the Features toolbar, are *also* on the new Mold toolbar. Other tools were moved to a new toolbar. For example, 3D Curve and Composite curve are now on the new Curve toolbar instead of the Reference Geometry toolbar. The following table displays:

- ☐ New tools (indicated by a check mark ✓)
- ☐ Additions to *existing* toolbars
- ☐ Existing tools that were moved to new or existing toolbars

Tool	✓	Tooltip	Equivalent Menu Selection
Note: The ✓ indicates that this is a <i>new</i> tool.			
Align toolbar			
	✓	Align Left	Tools, Align Annotations, Left
	✓	Align Right	Tools, Align Annotations, Right
	✓	Align Top	Tools, Align Annotations, Top
	✓	Align Bottom	Tools, Align Annotations, Bottom
	✓	Space Across	Tools, Align Annotations, Space Across
	✓	Space Down	Tools, Align Annotations, Space Down
Annotation toolbar			
	✓	Block	Insert, Annotations, Block
	✓	Model Items	Insert, Models Items
	✓	Stacked Balloon	Insert, Annotations, Stacked Balloon
Curves toolbar			
	✓	Projection	Insert, Curve, Projected
	✓	Split Line	Insert, Curve, Split Line
		Composite Curve	Insert, Curve, Composite
	✓	Curve Through Free Points	Insert, Curve, Curve Through Free Points
	✓	3D Curve	Insert, Curve, Curve Through Reference Points
	✓	Helix	Insert, Curve, Helix/Spiral

Tool	√	Tooltip	Equivalent Menu Selection
Note: The √ indicates that this is a <i>new</i> tool.			
Drawing toolbar			
	√	Dimension	View, Hide/Show Dimensions
	√	Show/Hide	None
	√	Crop View	Tools, Crop View
Layer toolbar			
		Layer Properties	None
Mold Tools toolbar			
	√	Cavity	Insert, Features, Cavity
		Scale	Insert, Features, Scale
		Draft	Insert, Features, Draft
	√	Split Line	Insert, Curve, Split Line
		Offset Surface	Insert, Surface, Offset
	√	Radiate Surface	Insert, Surface, Radiate
	√	Knit Surface	Insert, Surface, Knit
	√	Planar Surface	Insert, Surface, Planar
Selection Filter toolbar			
	√	Filter Surface Bodies	None
Sheet Metal toolbar			
		Insert Bends	Insert, Features, Sheet Metal, Bends
		Flattened	None
		No Bends	None
		Rip	Insert, Features, Sheet Metal, Rip
Sketch Tools toolbar			
	√	Polygon	Tools, Sketch Entity, Polygon
	√	Text	Tools, Sketch Entity, Text
	√	Construction Geometry	Tools, Sketch Entity, Construction Geometry
	√	Intersection Curve	Tools, Sketch Tools, Intersection Curve
Standard toolbar			
		Delete	Edit, Delete

Tool	✓	Tooltip	Equivalent Menu Selection
Note: The ✓ indicates that this is a <i>new tool</i>			
Surfaces toolbar			
	✓	Extruded Surface	Insert, Surface, Extrude
	✓	Revolved Surface	Insert, Surface, Revolve
	✓	Swept Surface	Insert, Surface, Sweep
	✓	Lofted Surface	Insert, Surface, Loft
	✓	Offset Surface	Insert, Surface, Offset
	✓	Radiate Surface	Insert, Surface, Radiate
	✓	Knit Surface	Insert, Surface, Knit
	✓	Planar Surface	Insert, Surface, Planar
	✓	Imported Surface	Insert, Surface, Imported
View toolbar			
	✓	Fast HLR	View, Display, Use Faceted HLR/HLG HLG - Hidden In Grey HLR - Hidden Lines Removed See Display Performance on page 1-15.
	✓	Display HLR in Shaded Mode	View, Display, HLR Edges in Shaded Mode

NOTE: Tools that display on your toolbar are based on the settings in your Windows Registry. You may need to customize your toolbars to include some of the new tools on new or existing toolbars.

Document Templates

Templates are documents (parts, drawings, and assemblies) that include user-defined parameters. These parameters are set in **Tools, Options, Document Properties**. See **SolidWorks Options** on page 1-9.

Templates allow you to maintain as many different documents for parts, drawings or assemblies as you need. A template can be a blank document, or it can be a part, drawing, or assembly that you saved as a template. For example, you can create:


- ☐ A document template using millimeters and another template using inches
- ☐ A document template using ANSI and another template using ISO dimensioning standard
- ☐ A base part in a document that you use for mold design

Opening a New Part

When you open a new part, drawing, or assembly, the **New SolidWorks Document** dialog box appears. The dialog box includes:

- ☐ **General tab** - part, drawing, and assembly templates
- ☐ **Preview area** - display a preview of the document
- ☐ **View icons** - display viewing configurations for the contents of any existing tabs

To create a template:

- 1 Click **New**  on the Standard Toolbar, or click **File, New**.
The **New SolidWorks Document** dialog box appears with the **General** tab active.
- 2 Double-click the **Part** icon (or the **Drawing** icon for a drawing template, and the **Assembly** icon for an assembly template).
- 3 Click **Tools, Options**. On the **Document Properties** tab, customize the options for your document template.

NOTE: Only options on the **Document Properties** tab are saved in a document template.

- 4 Click **File, Save As**.
- 5 In the **Save as type** list, select:
 - **Part templates** - (*.prtdot)
 - **Drawing templates** - (*.drwdot)
 - **Assembly templates** - (*.asmdot)
- 6 Type a name in the **File name** box. The extension is added automatically.
- 7 Browse to the folder where you want to save your template, then click **Save**.

Using Tabs

In addition to the **General** tab, you can create additional tabs. You can store any template under the **General** tab, or in any of the additional tabs that represent folders you created. Custom tabs help manage your documents in the following ways:

- ☐ **Placement** - Create folders on any drive, including hard drives, floppy drives, or network drives.
- ☐ **Access** - Share the contents of network folders, or restrict access to folders if required.
- ☐ **Organization** - Arrange folder contents to reflect your work environment.

To create a tab:




- 1 Using Windows Explorer, create a new folder in the location you want.
For example, the **General** folder exists on the following path: *installation directory\data\templates\...*
- 2 Open SolidWorks 2000 and click **Tools, Options, System Options, File Locations**.
- 3 In the **Show folders for** list, select **Document Templates**.
- 4 Click **Add** and browse using the **Choose Directory** dialog box to find the location of the folder you created in step 1.
- 5 Click **OK**.

A new tab appears in the **New SolidWorks Document** dialog box named after the folder you specified.

NOTE: First, you must create the folder with Windows Explorer. You can create the tab either before or after you create the templates.


Select the appropriate view:

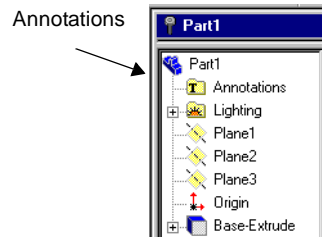
The three icons open views for either the **General** tab, or any custom tab you add.

- ☐ Select  to display items by using large icons as in Windows Explorer.
- ☐ Select  to display items by using small icons as in Windows Explorer.
- ☐ Select  to display information about each item in the window as in Windows Explorer, including columns for file size and date modified.

FeatureManager Design Tree

Annotations

The FeatureManager design tree now displays Annotations  below the part name. Previously, Annotations was the last item in the FeatureManager design tree. Functionality has not changed.



Duplicate Panel Display


You can now display a split instance of the panel adjacent to the graphics area, usually the FeatureManager design tree. A split display is *not limited* to duplicate FeatureManager design trees. You can select *any combination* of the following:

- ☐ FeatureManager design tree
- ☐ PropertyManager
- ☐ ConfigurationManager
- ☐ Third party applications that use the panel


This option is available either alone or in conjunction with **Window, New Window**. Without opening a new window, you can display the *same* part, assembly or drawing, along with any combination of the panels. With complex designs, for example, you can:

- ☐ Display different sections of the part, drawing, or assembly, expanded or collapsed
- ☐ View different details for configurations
- ☐ Pick different selections from each panel

To split the FeatureManager design tree:

- 1 Place the cursor at the top of the FeatureManager design tree until it changes to .
- 2 Drag the bar down below the last item in the FeatureManager design tree.
- 3 Release the mouse to display a duplicate FeatureManager design tree, below the first.

To close the split FeatureManager design tree:

- 1 Place the cursor at the top of the duplicate FeatureManager design tree until it changes to .
- 2 Drag the bar up until it meets the top of the original FeatureManager design tree.
- 3 Release the mouse to close the display of the duplicate FeatureManager design tree.

PropertyManager

Many functions in the SolidWorks 2000 application, such as Rib and Surfaces, use the PropertyManager instead of a separate dialog box. Using the PropertyManager means that your graphics are always displayed, rather than hidden by a dialog box. Previews, such as the direction of the rib feature or the surface extend, appear in the graphics area.

This functionality is independent of selecting the **Enable PropertyManager** check box in **Tools, Options, System Options, General**.

Shortcut Menu Functions

Using the shortcut menus (right-mouse button), you can access several new functions in the SolidWorks 2000 application. The functions themselves are *not* new, but they are now accessible from the shortcut menu.

ConfigurationManager

When you hold down the right mouse button in a blank area with the ConfigurationManager active, you can access **Add Configuration** and **Options** directly.

Add Configuration - opens the **Add Configuration** dialog box where you can create a new configuration for the existing part or assembly.

Document Properties - opens the **Document Properties** tab from the **Options** dialog box directly while the ConfigurationManager is displayed.

Additional Shortcut Menu Options

You have additional shortcut menu functions, if you prefer using the right mouse button rather than the menu bar. These added functions appear where appropriate. They include:

- ☐ **Delete** - delete a feature or portion of sketch
- ☐ **Suppress/Unsuppress** - suppress or unsuppress a feature or a component
- ☐ **Edit Equation** - edit an equation when you select the driven dimension in the sketch
- ☐ **Delete** - delete a Bill of Materials (BOM) in a drawing
- ☐ **Open** - open a part file or a top-level assembly over drawings
- ☐ **Mate** - mate components in an assembly
- ☐ **Move Component** - move a component in an assembly

Middle Mouse Button Functions

With a three-button mouse, you can dynamically use the following view commands:

- ☐ Pan all document types - Click the middle mouse button and hold down **Ctrl**.
- ☐ Rotate part or assembly - Click the middle mouse button.
- ☐ Zoom all document types - Click the middle mouse button and hold down **Shift**.

NOTE: In an active *drawing*, to *pan* you can use the middle mouse button with or without holding down **Ctrl**.

If you use a three button mouse, you may need to install the appropriate software or configure the device through Windows Program Manager. Consult the documentation included with your mouse.

SolidWorks Options

The structure and appearance of the Options dialog box have been enhanced so you can find the options more easily. The following sections describe the new structure of the dialog box.

Options Dialog Box Structure

The SolidWorks 2000 Options dialog box structure emphasizes the difference between system options and document properties. In previous releases, you selected whether to apply an option to the system defaults (for all new documents), the active document (only to the current document), or all possible (both current and new documents).

The Options dialog box now has only two tabs:

System Options - The system options are stored in the registry and are not part of the documents. Therefore, these changes affect all documents, current and future.

Document Properties - The document properties apply only to the current document, and the **Document Properties** tab is available only when a document is open. New documents get their document settings (such as Grid, Edge Display, Units, and so on) from the document properties of the template used in creating the new document.


The options listed on each tab are displayed in tree format. As you click an item in the tree, the options for the item appear as a page to the right of the tree window. The title bar reflects the title of the tab plus the title of the page.

Some pages are available only for certain types of documents. For example:

- **Material Properties** is available only when a part document is open.
- **Model/Feature Colors** is available only when a part or assembly document is open.
- **Routing** (formerly Piping) is available only when the SolidWorks Piping add-in is selected.

Accessing the Options Dialog Box

The proper tab and page of the Options dialog box is active when you access options as follows:

- Click **Tools, Options**. The **Options** dialog box appears with the **System Options** tab active.
- Right-click in the FeatureManager design tree area and select **Document Properties**. The **Options** dialog box appears with the **Document Properties** tab active. (In previous releases, the menu item was **Options**.)
- Click **Grid**  on the Sketch toolbar. The **Options** dialog box appears with the **Grid/Snap** page of the **Document Properties** tab active.

New Options Structure

Following is the new Options tree structure.

System Options	Document Properties
General	Detailing
Drawings	Dimensions
Default Edge Display	Notes
Area Hatch	Balloons
Colors	Arrows
Sketch	Virtual Sharps
Edge Display/Selection	Annotations Display
Performance	Grid/Snap
External References	Units
Default Templates	Colors
File Locations	Material Properties
FeatureManager	Line Font
Spin Box Increments	Image Quality
View Rotation	
Backups	
Routing	

Changes in Options

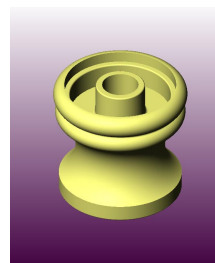
The tabs in the Options dialog box are now changed to a tree structure with pages. Some of the topics are restructured. For example:

- **Grid/Units** is now two separate pages: **Grid/Snap** and **Units**.
- **Detailing** is grouped on several different pages.
- **Crosshatch**, which was a separate tab in SolidWorks 99, is now on the **Material Properties** page.

Template locations - The **Default Templates** page is a new options category for specifying the folder and template file for parts, assemblies, and drawings imported from other applications.

Gradient colors - The **Colors** page now has a new check box, **Use gradient view background in parts and assemblies**. You can edit the colors for **Top Gradient Color** and **Bottom Gradient Color** in the **Colors** list to produce a background effect as shown.

Color schemes - Another new color feature is the ability to set up your own color schemes. You name a scheme, then edit a set of system colors to go with the name. The names are then available in a list, so you can choose a scheme from the list for any document. For more information, see the *SolidWorks 2000 Online User's Guide*.



SolidWorks Explorer

The SolidWorks Explorer is accessible from within the SolidWorks 2000 application. You can also open SolidWorks Explorer independently (by creating a shortcut on your desktop), and then open any SolidWorks documents from within SolidWorks Explorer.

To open SolidWorks Explorer from within the SolidWorks 2000 application, click **Tools, SolidWorks Explorer**. SolidWorks Explorer allows you to:

- ☐ View document dependencies for drawings, parts, and assemblies, using a tree structure.
- ☐ Copy, rename, or replace referenced documents. You have the option to find and update references to documents.
- ☐ View data and previews, or input data, according to the function you have active.

Use the following tools in SolidWorks Explorer to perform these tasks:



View any SolidWorks document on your system using the **Preview** tool. It is similar to using the shortcut menu in Windows Explorer for a **Quick View**.



View and edit specific, custom, or configuration-specific properties using the **Properties** tool. It also displays system data such as when the document was created and when it was last saved.



List all external references for SolidWorks documents using the **Edit References** tool.



Locate *assemblies*, *drawings*, or *parts* that reference a designated SolidWorks document using the **Where Used** tool.



Search for SolidWorks documents that meet a specified criterion, such as a custom property value, using the **Properties Search** tool.



List, rename, or delete configurations in an assembly or a part document using the **Edit Configurations** tool.



List, edit, and open any existing hyperlinks in a SolidWorks document using the **Edit Hyperlinks** tool.

Import and Export

Import Options

There is a new import option for IGES[®], STEP, ACIS[®], and VDAFS. You can now choose the **Try knitting surfaces** option. When selected, the SolidWorks software attempts to knit the surfaces in the imported file, but does not try to form a solid body from the surfaces.

To set import options:

- 1 Click **File, Open**.
- 2 In the **Files of type** list, click IGES, STEP, ACIS, or VDAFS.
- 3 Click **Options**.
- 4 Under **Knitting options**, select one of the following:
 - **Try forming solid model(s) from surfaces** - an attempt is made to knit the imported surfaces into a solid body
 - **Try knitting surfaces** - an attempt is made to knit the imported surfaces into surface bodies
 - **Don't knit at all** - no attempt is made to knit any of the imported entities
- 5 Click **OK**.

DXF/DWG Files

Changes to the way DXF and DWG files are handled include:

- ☐ **Insert, Sketch from Drawing**, has been removed from the SolidWorks software. Now that you can copy and paste sketch entities from a drawing to a part or assembly sketch, **Sketch from Drawing** is obsolete.
- ☐ You can export different assembly components in a drawing to different layers. Make sure you place the components on layers in the SolidWorks drawing prior to creating the DXF or DWG file.
- ☐ Spline entities and entities in paper space are now supported for import.

Parasolid Files

The **Files of type** list in the **Open** dialog box now includes **.xmt_txt**, Parasolid text files, and **.xmt_bin**, Parasolid binary files.

IGES Files

Enhancements to the IGES translator include:

- ☐ You can export or import all 2D sketch entities. In earlier versions, you could only import sketch entities on the XY plane and you could not export sketch entities. To export sketch entities, you must select the **Export sketch entities** check box in the **IGES Export Options** dialog box.
If the software detects 2D curves in the imported IGES file, the **IGES-In Surfaces, Curves, and Levels** dialog box appears. To import the curves as sketch entities, select the **Convert 2D curves into sketch entities** check box.
- ☐ You can export 3D curves or 3D sketch entities using the IGES translator. You can also import 3D sketch entities, as 3D curves, into a SolidWorks document.
- ☐ You can select a coordinate system to apply for export of 2D or 3D sketch entities.
- ☐ You can export both surfaces and solids in the same IGES file. The surfaces are marked as blanked surfaces in the IGES file to distinguish them from the trimmed surfaces used to create the solid.
- ☐ **MULTICAD** is now available as an export option for IGES output with trimmed surfaces. Select **MULTICAD** when you want to include coordinate system and attribute information in the IGES file.

STEP Files

Changes to the way STEP files are handled include:


- ☐ You can export multiple, disjoint surface bodies. In earlier versions, you were only able to export a single surface body in a STEP file.
- ☐ You can export STEP Application Protocol AP214 entities. Only the geometry, topology, and configuration information of the SolidWorks model is exported. Select **STEP AP214 (*.step)** from the **Save as type** list in the **Save As** dialog box.
- ☐ You can export configuration-controlled data in a STEP AP203 or AP214 file. Such data includes part names, release dates, revision numbers, and individuals and organizations who approved the releases.

To include configuration-controlled data in a STEP file:

- 1 With a part or assembly open, click **File, Save As**.
- 2 Select either **STEP AP203 (*.step)** or **STEP AP214 (*.step)** from the **Save as type** list.
- 3 Click the **Set STEP configuration data** check box.
The **STEP Configuration Data for Export** dialog box appears.
- 4 Select an item from the **Configuration Data Items** list then edit the item's properties in the **Edit Item** box. The properties that appear in the box are specific to the selected item.
- 5 To add an item:
 - a) Select an item in the **Configuration Data Items** list.
 - b) Select an item from the **Add Item** list, then click **Add**.
 - c) Edit the new item's properties as shown in Step 4.
- 6 Click **OK** to finish saving the STEP file with the configuration information.

Display Performance

Two display performance enhancements, especially relevant to complex parts and assemblies, were added to the SolidWorks 2000 application.

- ❑ **Dynamic transparency** - When you rotate a part or assembly with high-quality transparency, the application dynamically switches to a low-quality transparency, enabling you to rotate the model faster. Once you complete the rotation, the model switches back to high-quality transparency. You can set transparency in **Tools, Options, System Options, Performance, Transparency quality**.
- ❑ **Fast HLR** - Click **Fast HLR** , or **View, Display, Use Faceted HLR/HLG**, to implement the benefits of this new functionality. Although more apparent when viewing complex models, you can select **Fast HLR** with any model to see the following benefits:
 - **Real time rotation** - You can now control the speed of rotation and display the complete model in real time. In prior releases, part of the model remained hidden until you released the mouse button.
 - **Dynamic HLR recalculation** - HLR now displays *during rotation*, instead of only when you release the mouse button, as it did in previous releases.
 - **Resolution and performance** - Image resolution and performance are dependent on the settings in **Options, Document Properties, Image Quality**. You can balance image quality (resolution) with performance.


Sketches

There is new 2D and 3D functionality in SolidWorks 2000. This chapter discusses the following:

- ☐ New Sketch Toolbar tools
- ☐ Parametric and editable sketch patterns
- ☐ Splines
- ☐ 3D Sketching


New Sketch Toolbar Tools

Text Tool

You can now select **Text**  from the Sketch Tools toolbar to sketch text on a model face. This functionality is still available on the **Tools, Sketch Entity** menu. The functionality has not changed from SolidWorks 99.



Construction Geometry

You can now select **Construction Geometry**  from the Sketch Tools toolbar to convert sketched entities to construction geometry, and to convert construction geometry to sketched entities. The **Construction Geometry** tool is available while working in a sketch of a part, in an assembly, or in a drawing document.




As in earlier releases of SolidWorks, it is also possible to convert sketch entities, as follows:

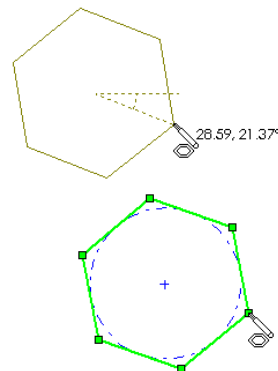
- When a drawing is active, by clicking **Tools, Sketch Entity, Construction Geometry**.
- By selecting a sketch entity in a sketch, an assembly, or a drawing document and then clicking the **For construction** check box on the PropertyManager tab.

Polygon

There is a new **Polygon** tool  on the Sketch Tools toolbar and on the **Tools, Sketch Entity** menu that provides an easy way to sketch polygons.

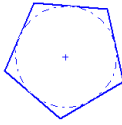
To create a polygon:

- 1 In an open sketch, click **Polygon**  or click **Tools, Sketch Entity, Polygon**. The pointer changes to .
- 2 Use the spin box  on the PropertyManager to specify the number of sides before you sketch the polygon. Or you can change the number of sides after you sketch the polygon.
- 3 Point where you want to place the center point of the polygon, and drag out the polygon to the size that you want.

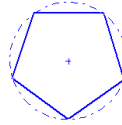


- 4 Click the **Inscribed circle** check box to display an inscribed circle. Click to clear the check box to display a circumscribed circle

Inscribed
circle



Circumscribed
circle



- 5 You can edit the following polygon values in the PropertyManager:



Change the number of sides using the spin box.



Change the X and Y coordinates of the center point.




Modify the diameter of the inscribed or circumscribed circle.



Change the angle of rotation.




Lock the number of sides for the current polygon. Any change that you make to the number of sides is used for the *next* new polygon that you create.

- 6 To change the position, rotation, or size of the polygon by dragging, click **Select** .
- Resize the polygon by dragging it by a side.
 - Move the polygon by dragging one of its vertices.

To edit a polygon:

- 1 Edit the sketch containing the polygon.
- 2 Select one of the sides of the polygon and click **Tools, Sketch Tools, Edit Polygon**.
- or -
Right-click a side of the polygon and select **Edit Polygon**.
- 3 Modify the polygon by dragging, as described in step 6 above, or by using the PropertyManager.

Intersection Curve



The new **Intersection Curve** tool  on the Sketch Tools toolbar opens a sketch and creates a sketched curve at the following kinds of intersections:

- A plane and a surface or a model face
- Two surfaces
- A surface and a model face
- A plane and the entire part
- A surface and the entire part


You can use the resulting sketched intersection curve in the same way that you use any sketched curve, including the following tasks:


- Measure thickness at various cross sections of a part
- Create sweep paths that represent the intersection of a plane and the part
- Make sections out of imported solids to create parametric parts

To use the sketched curve to extrude a feature, the sketch that opens must be a 2D sketch. Other tasks can be performed using a 3D sketch.

- To open a 2D sketch, select the plane first then click **Intersection Curve** .
- To open a 3D sketch, click **Intersection Curve**  first then select the plane.

To measure the thickness of a cross section of a part:

- 1 Open **Mouse_cover.sldprt**.
- 2 Click **Intersection Curve**  or **Tools, Sketch Tools, Intersection Curve**.

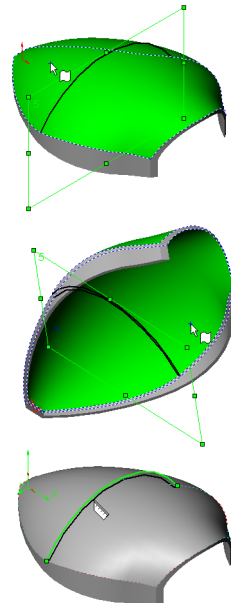
A 3D sketch opens (because you clicked  first).

- 3 Select the intersecting items:
 - Select **Plane5**.
 - Click the face of the part.

A sketch spline appears at the intersection.

- 4 Rotate the part, then click the opposite face of the part.
Another sketch spline appears on the second face.

- 5 Now you can select **Tools, Measure** and measure the distance between the two sketched splines.



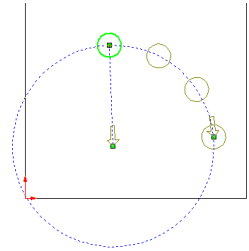
Parametric and Editable Sketch Patterns

Sketch patterns are now parametric. Additionally, you can now edit sketch patterns after completion.

Circular Sketch Step and Repeat

In SolidWorks 2000, the construction of the circular sketch pattern has changed.

- ☐ You can now position the center point of the circular pattern by entering values for the X and Y coordinates of the center point of the pattern arc.
- or -
You can drag the center point of the pattern arc to a location that you select.
- ☐ Use the **Constrain spacing** check box to define the angle of the circular pattern. After the pattern is created, you can change this angle in the same way that you change any other dimension. (This option is only available if the **Equal** check box is not checked.)
- ☐ There is a new **Fixed** option that is related to the **Radius** value. If you select the **Fixed** check box before you create the pattern, the result is as follows:
 - The radius is displayed as an explicit dimension.
 - You can control the radius by double-clicking the dimension and using the **Modify** dialog box.



To edit a circular sketch pattern:

NOTE: Before editing the circular sketch pattern, it is a good idea to add relations or dimensions to the initial feature of the pattern.

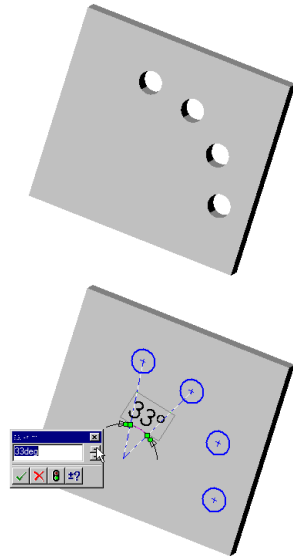
- 1 To edit the circular pattern feature, right-click the pattern sketch and select **Edit Sketch**.
- 2 You can modify the pattern in the following ways:
 - Double-click the angle dimension and change the angle in the **Modify** dialog box.
 - Drag the center point of the pattern to a new location.
 - Change the rotation of the pattern by dragging the first instance of the pattern by its center point or vertex.
 - Change the radius of the pattern arc by dragging the first instance of the pattern by its center point or vertex.

To make the pattern radius *larger*, drag it away from the center of the pattern arc.

To make the pattern radius *smaller*, drag it toward the center of the pattern arc.
 - Select and delete individual pattern instances.
- 3 To change the number of pattern instances, select an instance and click **Tools, Sketch Tools, Edit Circular Step and Repeat**.

- or -

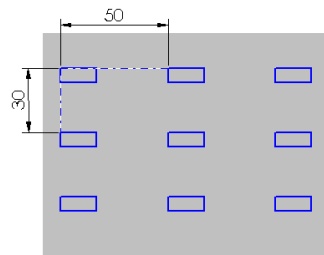
Right-click an instance and select **Edit Circular Step and Repeat** from the shortcut menu.
- 4 Change **Step, Number** and click **OK**.
- 5 Exit the sketch to complete the new pattern feature.



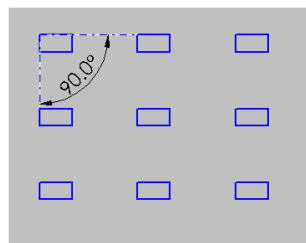
Linear Sketch Step and Repeat

- ❑ There is a new **Fixed** option in the **Linear Sketch Step and Repeat** dialog box that applies to the **Spacing** value for each linear direction. If you select the **Fixed** check box, and then create the pattern by clicking **OK**, the result is as follows:
 - Corresponding points on the first and second instance are joined with a construction line, and this line is dimensioned.
 - The resulting dimension is displayed as an explicit sketch dimension.
 - You can control the spacing by double-clicking the dimension and using the **Modify** dialog box.
- ❑ The new **Constrain angle between axes** check box is available if you define two pattern directions. When you select **Constrain angle between axes**, the angle is defined by a construction line through corresponding points on the first and second instance, in each direction. After the pattern is created, by clicking **OK**, you can change this angle in the same way that you change any other dimension.

The spacing is fixed.



The angle between axes is constrained.



To edit a linear sketch pattern:

NOTE: Before editing the linear sketch pattern, it is a good idea to add relations or dimensions to the initial feature of the pattern.

- 1 To edit the linear pattern feature, right-click the pattern sketch and select **Edit Sketch**.
- 2 You can modify the pattern in the following ways:
 - Drag a point or vertex on one of the pattern instances.
 - Change the angle of the pattern by double-clicking the angle and changing its value in the **Modify** dialog box.
 - Add dimensions and change their values using the **Modify** dialog box.
 - Add relations to the pattern instances.
 - Select and delete individual pattern instances.
- 3 To change the number of pattern instances, select an instance and click **Tools, Sketch Tools, Edit Linear Step and Repeat**.
 - or -
 - Right-click an instance and select **Edit Linear Step and Repeat** from the shortcut menu.


- 4 Change the **Number** in either one or both directions, and click **OK**.
- 5 Exit the sketch to complete the new pattern feature.

Splines

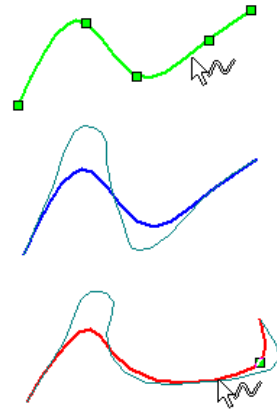
Curvature Evaluator for Splines

The new curvature evaluator tool provides visual feedback of the slope and curvature of the spline curve. You can use the curvature evaluator in a part or drawing document.

To evaluate the curvature of a spline:

- 1 In a sketch, click **Select**  and right-click the spline. (To use the curve evaluator with more than one curve, hold down **Ctrl** while you right-click two or more curves.)
- 2 Select **Inspect Curvature** from the shortcut menu.
 - A *comb* appears that displays the inflection of the curve. The locations where the comb crosses the spline are “points of inflection.”
 - As you manipulate the spline, the comb changes dynamically.

To dismiss the comb, right-click the spline again, and select **Remove Curvature Information**.



Relations for Splines

You can now specify **Perpendicular** and **Tangent** relations for through points on a spline.

You can now add a tangent relation between a the endpoint of a spline and the endpoint of an arc, a line, a parabola, or another spline.

Relations can be added for spline points in 2D and 3D sketches.

The Moving Frame

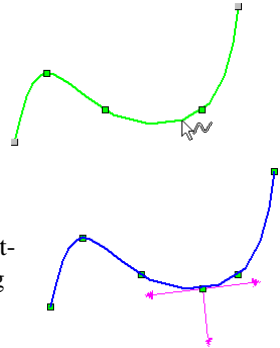
The *moving frame* provides you with a way to directly manipulate a spline without making the spline more complex by adding points to the spline. You can use the moving frame in a part or drawing document.

NOTE: The moving frame can be used with splines created in SolidWorks that you are able to modify. It cannot be used with imported or converted splines.

To add a moving frame to a spline:

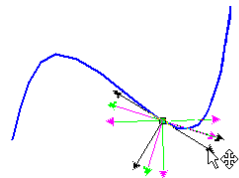
Right-click the sketched spline and select **Moving Frame** from the shortcut menu. You can use the moving frame in the following ways:

- Drag the moving frame by the frame point to move it along the spline to wherever you want to place it.
- Lock the moving frame into the chosen position by right-clicking the midpoint of the moving frame and selecting **Lock Moving Frame On Curve**. To unlock, right-click and select **Unlock Frame**.



NOTE: When you unlock the frame, any relations are also deleted.

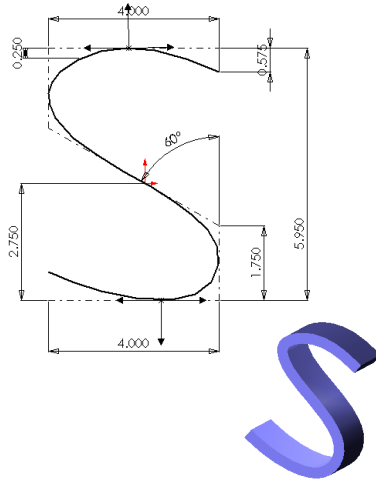
- Pull the frame handles to change the shape of the spline curve.
- Add multiple moving frames to a spline, making it possible to control the curve of the spline where there are no spline points.
- Select the frame and modify its angle or X and Y coordinates by entering new values in the PropertyManager.



- Add **Tangent** or **Coincident** relations between the frame and an entity or a model edge.

The graphic shows the sketch of a fully defined spline, and the resulting thin feature extrusion.

- Remove a moving frame by selecting it and pressing **Delete**.



PropertyManager Display

PropertyManager symbols dynamically describe the spline and the moving frame's location while you drag the moving frame along the spline:



Displays the X, Y, and Z coordinates of the frame point






Describes the curvature of the spline as the frame moves




Describes the angle of tangency along the X, Y, and Z coordinates or, in a 3D sketch, in the XY, YZ, and XZ planes


3D Sketching

New functionality in SolidWorks 2000 sketching includes the following:

- The PropertyManager is now available in 3D sketching.
- You can use **Trim**  and **Extend**  from the Sketch Tools toolbar.
- You can use the **Convert Entities** tool  to convert a model edge, loop, face, external sketch curve, and so forth, for use in a 3D sketch.

Splines in a 3D Sketch

You can now use **Spline**  to create splines in 3D sketches, and you can use the curvature evaluator tool described above.

- When creating splines in a 3D sketch, use the **Tab** key to change planes, as when you create lines. Use the **Ctrl** key to select a sketch plane based on a model face.
- You can use **Fillet**  to create an arc between two planar splines, or between a spline and a line, in a 3D sketch.
- You can add relations for spline points in 3D sketches.

Surfaces

This chapter describes enhancements to surfaces for the SolidWorks 2000 application. The topics include:


- ☐ Surface bodies
- ☐ Extend surface bodies
- ☐ Trim surfaces, single and mutual surface trim
- ☐ Fillet surfaces
- ☐ Delete a face from a surface
- ☐ Insert a curve on a surface
- ☐ Cut a solid using a surface body
- ☐ Knit surface changes
- ☐ Hide and show surface bodies


Surface Bodies

In earlier releases, using tools such as **Extrude** or **Loft**, you could create only base surface features. With the SolidWorks 2000 application, you can still use the same tools to create a base surface feature, but you now have the following additional capabilities:

- ☐ Modifying the surface entities
- ☐ Creating multiple surface entities
- ☐ Modifying the output of multiple surface entities

To identify these new capabilities, the term *surface body* is applied. A surface body has the characteristics of a surface, such as zero-thickness geometry, but a surface body includes *capabilities* that a surface does not have. These capabilities which modify the geometry of a surface body, such as **Extend** or **Trim**, are described in the remainder of this chapter.

You can select a surface body from the graphics area using **Filter Surface Bodies**  from the Selection Filter toolbar. You select all types of surfaces with **Filter Surface Bodies**. This includes a surface you created by extruding a line, or one that you created from a sketch and later manipulated using the tools such as **Trim**.

The cursor changes to , indicating when the pointer is over a surface body.

Surface bodies can include:

- ☐ Single face surfaces
- ☐ Multiple face surfaces
- ☐ Multiple surfaces that intersect
- ☐ Adjacent or connected surfaces
- ☐ Filleted surfaces

Extend Surfaces

You can extend a surface body, by selecting one or more edges, or by selecting one or more faces.

Definitions for Extend Surfaces

To extend surfaces, you must define the following in the PropertyManager:

- ☐ **Extend edge**
 - Select a single edge in the graphics area to extend that edge.
 - Select multiple edges in the graphics area to extend more than one edge.
 - Select the *face* in the graphics area to extend the surface along *all* of its outer edges.
 - Select *multiple faces* in the graphics area to extend the surface along all the outer edges of all faces.
- ☐ **Extension type**
 - Click **Same surface** to extend the surface along the geometry of the surface.
 - Click **Linear** to extend, using ruled surfaces, tangent to the original surface along the edges.
- ☐ **End condition**
 - Click **Distance** and enter a value in the **Distance** box.
 - Click **Up to point** and select a point in the graphics area, such as a vertex or a reference point to display in the **Reference point** box.
 - Click **Up to surface** and select another surface, a plane, or a solid part to display in the **Reference point** box.

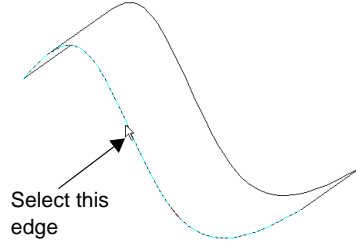
Characteristics and Capabilities of Extend Surfaces

With extend surfaces, use the PropertyManager to edit any of the following:

- ☐ **Edges** - Select a different edge, more than one edge, or the face of the surface body.
 - Single edge - You can use **Same surface** or **Linear**, and specify either **Distance**, **Up to point**, or **Up to surface**.
 - Multiple edges *or* face - You can use either **Extension type** and specify **Distance**.
- ☐ **Extension type** - You can modify the type.
- ☐ **End condition** - You can specify an end condition, based on **Extension type** and your selection of **Edges**.

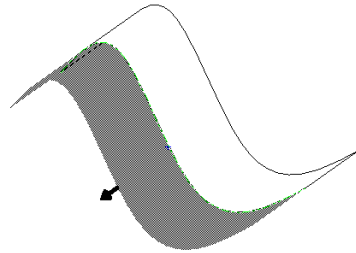
To extend a surface along an edge for a specified distance:

- 1 Open **Surface_extend_dist.sldprt**.
- 2 Select the long edge closest to you, as shown.
- 3 Click **Insert, Surface, Extend**.
- 4 In the PropertyManager, select the following:
 - a) Leave **Extension type** as **Same surface**.
 - b) Leave **End condition** as **Distance**



- c) Set **Distance** to 20mm.
- 5 Click **OK**.

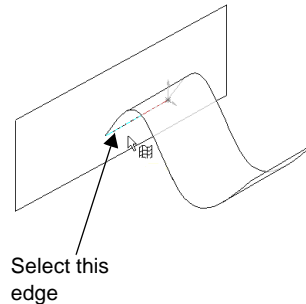
The surface is extended and the **Surface-Extend1** feature appears in the FeatureManager design tree



To extend a surface up to a surface:

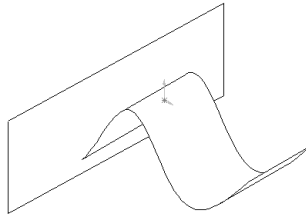
- 1 Open **Surface_extend_to_srf.sldprt**.
- 2 Select the short edge, that includes the Origin, as shown.
- 3 Click **Insert, Surface, Extend**.
- 4 In the PropertyManager, select the following:
 - a) Leave **Extension type** as **Same surface**.
 - b) Set **End condition** to **Up to surface**
 - c) In the graphics area, select the adjacent surface as the **Reference surface**.

If you rotate the part to get a different perspective, you will see the solid arrow preview while you rotate.



5 Click **OK**.

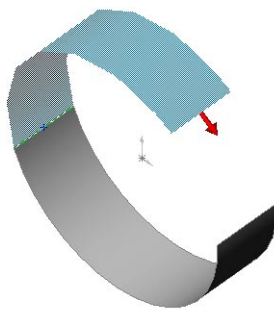
The surface is extended along the edge you selected up to the second surface (**Reference surface**), and **Surface-Extend2** appears in the FeatureManager design tree.



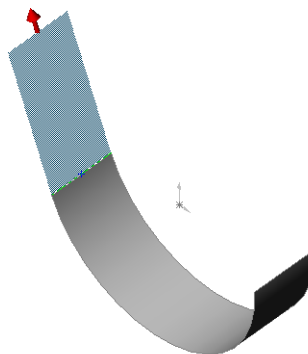
NOTE: If you extend a surface up to another plane, first select the plane in the FeatureManager design tree. Then click **Insert, Surface, Extend**. The plane is visible in the PropertyManager. Select it as the **Reference surface**.

Extend Surfaces Samples

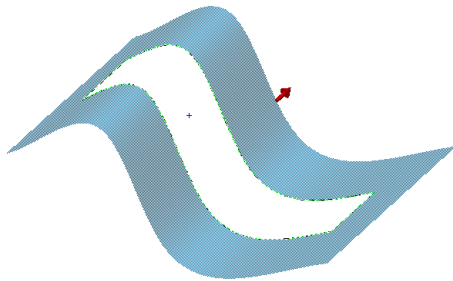
Below are examples of how you can manipulate surface bodies using **Extend Surface**.



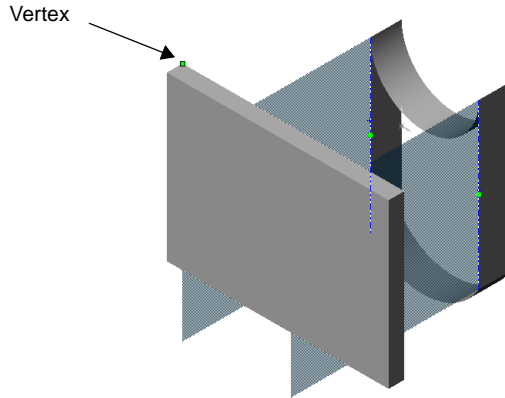
Extend Edge using Same surface
as the Extension type



Extend Edge using Linear
as the Extension type

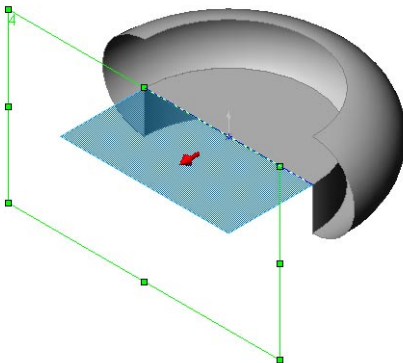


Extend Surface using a face
in the Extend edge box

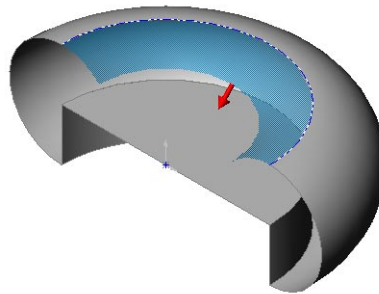


Extend Surface edges Up To point using a
vertex on the solid as the Reference Point

NOTE: When you select to extend the face of a surface (all edges), or multiple edges, only one arrow appears in the preview. The position of the arrow, depends the location of your pointer.



Extend Surface Up to point Using a
plane as the Reference Point



Extend Surface Using Same surface as the
Extension Type and Distance as the End condition

Trim Surface

You can use a surface as a trim tool to trim other surfaces where they intersect. You can also use a surface in conjunction with additional surfaces, as mutual trim tools.

Definitions for Surface Trim

To trim surfaces, you must define the following in the PropertyManager:

- ☐ **Surface** - Select one or more surfaces to trim
- ☐ **Trim Tool** - Select any of the following as a trim tool:
 - Surface - One surface as the tool to trim another surface
 - Plane - A plane as the tool to trim a surface
 - Sketch - The extrusion of the sketch, along the perpendicular direction of the sketch, as the trimming surface
- ☐ **Mutual Trim** - Intersecting surfaces to mutually trim the surfaces to each other

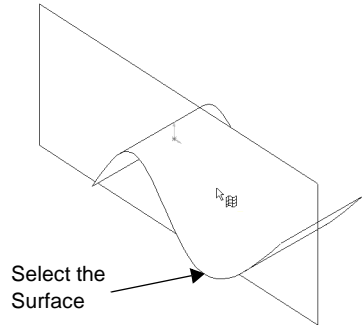
Characteristics and Capabilities of Surface Trim

With trim surfaces, use the PropertyManager to edit any of the following:

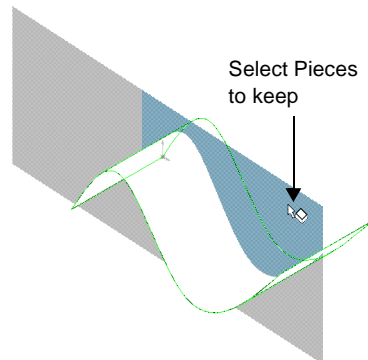
- ☐ **Boundaries** - The trim tool does not need to completely cross the boundaries of the surface you intend to trim. The surface is trimmed along the shortest line, drawn through the vertices of the intersection of the trimming tool, and the surface.
- ☐ **Edit** - You can edit trim definitions in the PropertyManager.
- ☐ **Knit Surfaces** - Selecting the **Mutual Trim** option, you can trim a surface into multiple pieces. You can select more than one piece. All the chosen contiguous pieces are knitted together. See **Knit Surfaces** on page 3-13.

To trim a surface:

- 1 Open **Surface_trim_single.sldprt**.
- 2 Click **Insert, Surface, Trim**.
- 3 In the PropertyManager, select the following:
 - a) Leave **Type** as **Trim Tool**
 - b) In the graphics area, select the surface **Surface-Extrude1**, as shown, as the **Trim Tool**.

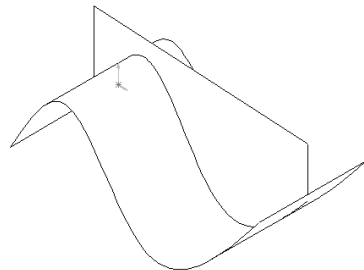


- c) In the graphics area, select the surface **Surface-Extrude2-Trim 1**, as shown, as the **Pieces to keep**.



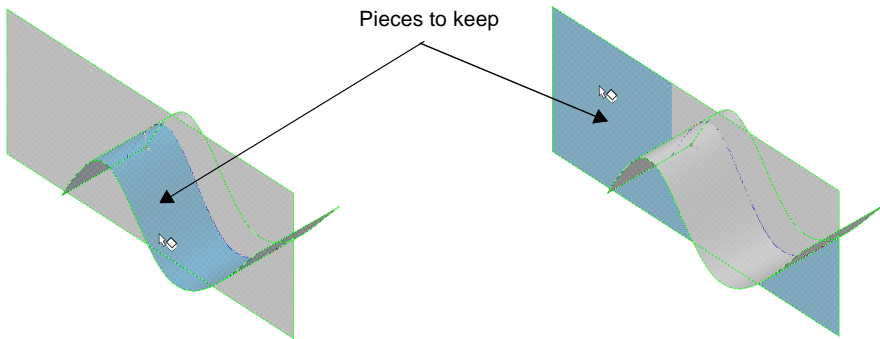
NOTE: With certain surface intersections, you can also select other portions to trim and achieve different results.

- 4 Click **OK**.
Surface-Trim1 appears in the FeatureManager design tree.



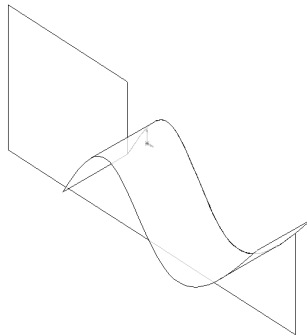
To trim a surface using mutual trim:

- 1 Open **Surface_trim_mutual.sldprt**.
- 2 Click **Insert, Surface, Trim**.
- 3 In the PropertyManager, select the following:
 - a) For **Type**, select **Mutual Trim**.
 - b) In the graphics area, select *both* surfaces as the **Trimming Surfaces**.
 - c) In the graphics area, select the two areas, as shown, as the **Pieces to keep**.



- 4 Click **OK**.

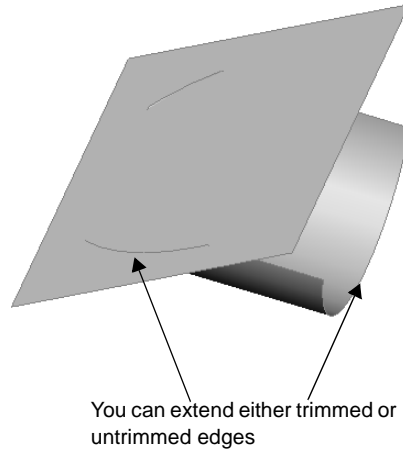
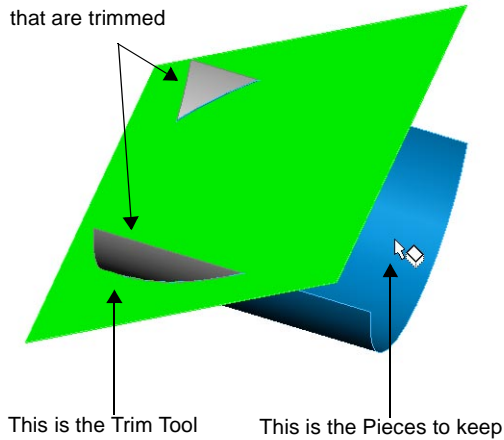
All the trimmed pieces chosen are knitted together. **Surface-Trim1** appears in the FeatureManager design tree.



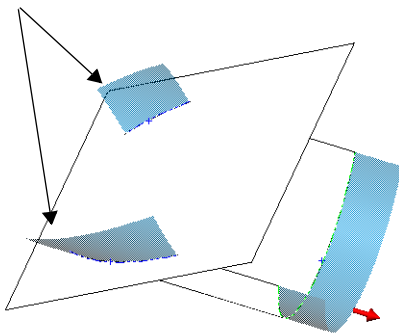
Trim and Extend Surfaces Samples

You can use all of the attributes of **Extend Surface** with any surface you trimmed using either **Trim Tool** or **Mutual Trim**. Below are some examples of how you can manipulate surface bodies using **Extend Surface**, after you use **Trim**.

These are the surfaces that are trimmed

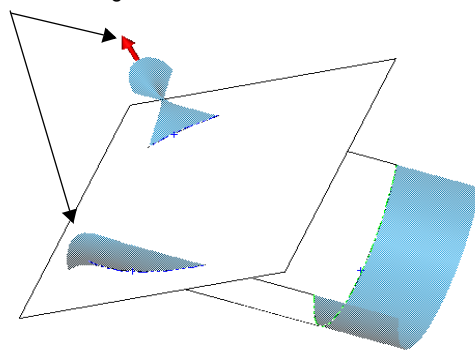


Trimmed edges extended



Surface Extend after Trim using Same surface as the Extension type with Distance as the End condition

Trimmed edges extended




Surface Extend after Trim using Linear as the Extension type with Distance as the End condition

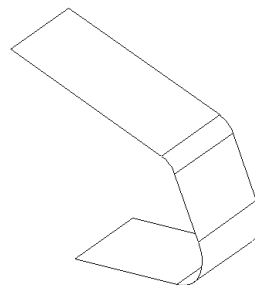
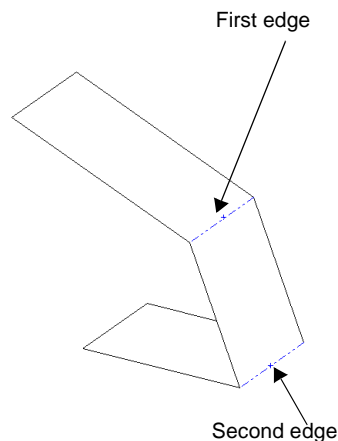
Fillet Surface

You can now fillet surface bodies in the following ways:


- ☐ Use a fillet to smooth the edge between two adjacent faces in a surface body that meet at an angle.
- ☐ Use a **Face Blend** fillet to combine multiple, separate surface bodies into a single surface body.
- ☐ Use a **Multiple radius fillet** to assign different radius values when you have multiple, adjacent surfaces. See **Multiple Radius Fillets** on page 4-12.

To fillet a surface using multiple radius fillets:

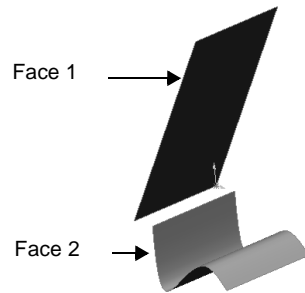
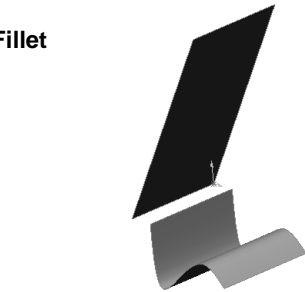
- 1 Open **Surface_fillet.sldprt**.
- 2 Click **Fillet**  or **Insert, Surface, Fillet/Round**. In the **Fillet Feature** dialog box, enter the following values:
 - a) Click the **Multiple radius fillet** check box.
 - b) Click to clear the **Propagate to tangent faces** check box.
- 3 In the graphics area, select the first edge, at the bottom end of the longer surface, as shown.
- 4 Set **Radius** to 5mm.
- 5 In the graphics area, select the second edge, as shown.
- 6 Set **Radius** to 15mm.
- 7 Click **OK**.



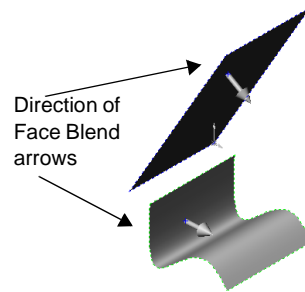
To create a face blend fillet between surfaces:

- 1 Open **Surface_multi_combo.sldprt**.
- 2 Click **Fillet**  or **Insert, Surface, Fillet/Round**. In the **Fillet Feature** dialog box, enter the following values:
 - a) In the **Radius** box, enter 15mm.
 - b) In the **Fillet type** list, click **Face Blend**.

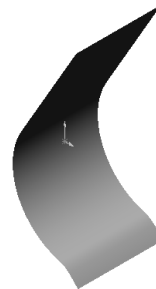
- c) In the graphics area, select Face 1, as shown, for **Face set 1**. The preview arrow indicates the direction of the face blend.
- d) If it points *away* from Face 2, click **Flip Side** to reverse the direction of the fillet.



- e) Click **Face set 2**, and select Face 2 in the graphics area, as shown. The arrow on Face 2 points *towards* the arrow on Face 1, as shown.

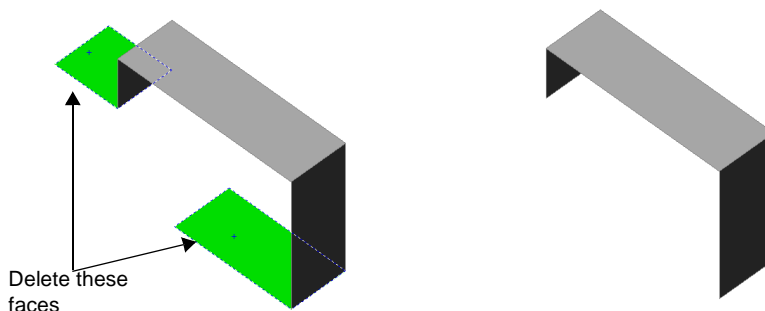


- 3 Click **OK**.
The **Face Blend** fillet combines the two separate face bodies into a single model.



Surface Delete

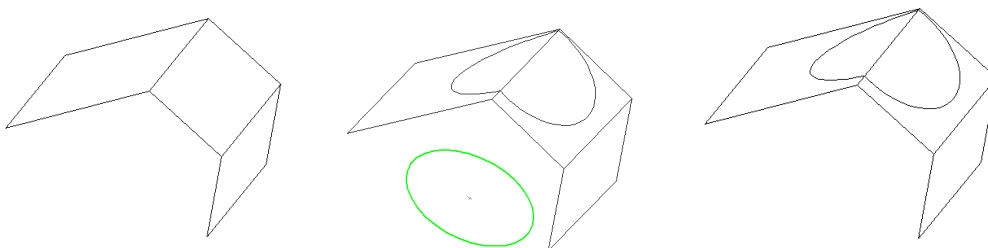
You can delete one or more faces of a surface by selecting the faces in the graphics area and pressing **Delete**. This displays the **Face Deletion** dialog box, where you select to either **Delete Face(s)** or **Delete Feature**.



Split Line on Surface

You can now project a sketched curve onto a body. In earlier versions, you could only create a Split Line on faces of a solid. Follow these steps to project a curve onto a surface:

- 1 Create the surface body.
- 2 Create a curve body on a different plane.
- 3 Click **Insert, Curve, Split Line**.



Knit Surfaces

The following applies to knit surface bodies in the SolidWorks 2000 application:

- ☐ With a knit surface, you can now select the entire surface body.
- ☐ Knit surfaces no longer absorb the surfaces you used to create them.

Hide and Show Bodies

You can now hide or show surface (or solid) bodies, using **View, Hide/Show Bodies**. This opens the PropertyManager, where you can select one or more surface bodies or a solid body from the graphics area.

- 1 Click **View, Hide/Show Bodies**.
- 2 Select one or more entities in the graphics area. This places your selections in the **Select Bodies** box.

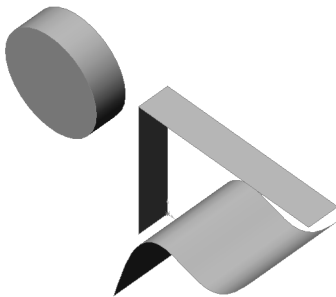
When you select a solid body or a surface body, it becomes translucent in the graphics area.

- 3 Click **OK** to close the PropertyManager.
The entities you selected remain hidden.
- 4 Click **View, Hide/Show Bodies** again to open the PropertyManager and select the translucent bodies in the graphics area.
- 5 Click **OK** to close the PropertyManager.

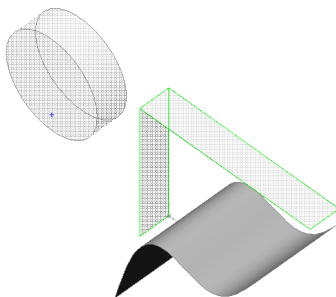
The translucent entities you selected are visible again.

You can also use the right mouse button in the graphics area to hide a surface without using the PropertyManager to preview the entities you select. To show the hidden surface again, click **View, Hide/Show Bodies** again, select the translucent bodies in the graphics area and click **OK**.

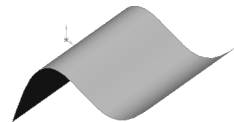
You can still select from the FeatureManager design tree to **Hide** or **Show**, but it does not open the PropertyManager and display the model in a translucent state.



View of two surface bodies and a solid body



Select Hide/Show Bodies to hide selected entities



Close PropertyManager, and the selected entities remain hidden.

Features and Parts

The 3D part is the basic building block of the SolidWorks mechanical design software. This chapter describes the enhancements to parts, including:

- ☐ Mass properties
- ☐ Hole Wizard
- ☐ Table-driven patterns
- ☐ Sketch-driven patterns
- ☐ Fillet features
- ☐ Rib features
- ☐ Shell features
- ☐ Non-uniform scaling

General Information

Derived Parts

You can select which configuration of the original part is used for a derived part. The three types of derived parts are: **Base Part**, **Mirror Part**, and **Derived Component Part**.

To set the configuration for a derived part:

- 1 Create the derived part.
- 2 In the derived part, right-click the top level part icon in the FeatureManager design tree and select **List External Refs**.
- 3 In the **External References For** dialog box, select the configuration to use in the **Use named configuration** list.

Equations

There is now an **Undo** button on the **New Equations** and **Edit Equations** dialog boxes.

Link Values

You can delete unused dimension names from the **Name** list in the **Shared Values** dialog box. Select the name and press **Delete**. You cannot delete a name used by another dimension.

Check Entity

Enhancements to the **Check Entity** functionality, which you access by clicking **Tools, Check**, include:

- ☐ **Check All** - You can choose to check the entire body, only the solid model, or only the surface bodies.
- ☐ **Check Selected items** - You can now select more than one entity (face, edge, or surface body) to check at the same time.
- ☐ **Maximum edge gap** and **Maximum vertex gap** - When selected, the software reports the maximum tolerance between the edges and vertices in the selected items.

Mass Properties

There are several improvements in the calculation of mass properties.

- ☐ **Density** - You can now change the density of a part from the **Measurement Options** dialog box. Click **Tools, Mass Properties**, then **Options** to access this dialog box.
- ☐ **Units** - When changing the density value, you can enter the value using any units. For example, if the units of the part are grams and millimeters, you can enter a density value using pounds and inches. The software converts the value to the document's units when you click **OK**.
- ☐ **Updates** - You can update the mass properties information when you save a document. The benefit is that the next time you access the mass properties, the system does not need to recalculate them (if the document has not changed). To set this option, click **Tools, Options, System Options, Performance** and click the **Update mass properties while saving document** check box.
- ☐ **Coordinate Systems** - When you calculate mass properties using a coordinate system, the moments of inertia are calculated at the origin of the coordinate system, using its axes.

Hole Wizard

The Hole Wizard in the SolidWorks 2000 application includes a redesigned user-interface. The new user-interface includes a completely reorganized dialog box, new parameters, dynamic updating, as well as many additional and more complex hole types.

Organization

The Hole Wizard uses an intuitive organizational structure. Instead of scrolling through an alphabetical list of hole types, each hole type is identified by a tab.

To facilitate implementation, each appropriate element in the **Hole Definition** dialog box, (such as **Hole Type and Depth**), is displayed, addressed, and previewed independently for each hole type. For example, when you select **Pipe Tap** as a hole type, only the elements appropriate to this hole type display in the dialog box.

Dynamic Updates

The properties, values, and hole preview update depending on which hole type you select. The values of the hole update based on the size of the fastener. The three dynamic update areas include:

- ☐ **Property** - Select a tab to display each hole type, and the appropriate changes are displayed in the **Property** column. For example, the **Property** column includes a listing for **Standard** as well as **Screw Type**.

- ☐ **Value** - The listings in the **Value** column also vary, depending on which hole type you select. Some display a retrieved characteristic, while others are user-defined. For example, all hole types include industry-standard, user-selected, definitions such as ANSI-Metric and JIS. However, if you select a **Tap**, as opposed to **Counterbore** hole type, the *different* and *appropriate*, user-defined parameters are retrieved.

NOTE: Values that do not use the document's units, use the units of the selected fastener type.

- ☐ **Preview** - The dynamic previews include a rendering of the hole type in its entirety, as well as detailed views of each appropriate element. All are displayed in contrasting colors, clear of the dimensions and parameters which you select in the **Value** column.

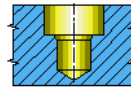
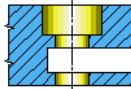
Favorites

You can now create, save, update, and delete your most commonly used hole types. This allows you to quickly apply any saved hole types to a SolidWorks document.

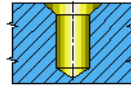
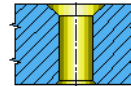
Hole Wizard Samples and Tabs

The following is a list of the new tabs and samples of the previews. These section previews display in the **Hole Definition** dialog box.

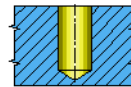
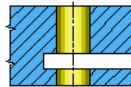
- ☐ **Counterbore** - You can create counterbored holes with a blind, through, or through all clearance hole.



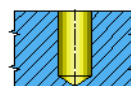
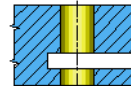
- ☐ **Countersink** - You can create countersunk holes with a blind, through, or through all clearance hole.



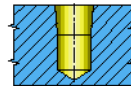
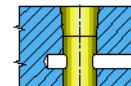
- ☐ **Hole** - You can create blind, through, or through all plain holes with or without near and far side countersinks.



- ☐ **Tap** - You can create blind, through, or through all tapped holes.



- ☐ **Pipe Tap** - You can create blind, through, or through all tapered pipe tapped holes.



- ☐ **Legacy** - You can edit the definition of holes created prior to SolidWorks 2000 using the parameters on this tab.

Table-Driven Patterns

Using X-Y coordinates, you can specify a feature pattern. Hole patterns using X-Y coordinates are a common application for table-driven patterns. However, you can use other features, such as a boss, with table-driven patterns. You can also save and retrieve the X-Y coordinates of a feature pattern (see page 4-6). Selecting a table-driven pattern can resolve situations in which you have:

- ☐ An irregular pattern
- ☐ An existing set of coordinates relative to the part or assembly

Definitions for Table-Driven Patterns

To create table-driven patterns, you must define the following:

- ☐ **Seed features** - Create seed features that represents the geometry you want to replicate.
- ☐ **Reference point** - Select a reference point as an origin. The origin defines what point of the seed feature corresponds to the X-Y coordinates you define in the table.
 - You can use the geometric center of the seed feature (centroid).
 - You can define a reference point, such as a sketch point or a vertex.
- ☐ **Coordinate system** - Create a coordinate system to act as the point of origin to define the X-Y coordinates that populate the table.
- ☐ **X-Y points** - Designate X-Y coordinates in the table, with each coordinate set representing an instance of the pattern. You can specify negative coordinates.

Centroid

If you select centroid as the reference point, the system determines the centroid based on the type of seed feature.

- ☐ For cylindrical, conical, or revolved features, the centroid is the intersection of the axis of revolution with the X-Y plane of the pattern.
- ☐ If the sketch consists of lines and arcs, such as rectangles or oval shapes, and if the sketch plane is parallel to the X-Y plane, the centroid is defined as the centroid of the sketch.
- ☐ For any other condition, the centroid is the centroid of the face of the seed feature.
- ☐ If there is more than one seed feature, the centroid of the first seed feature is used.
- ☐ If the seed feature is generated from a sketch that includes multiple contours, the centroid of the largest *closed* contour is used.

Characteristics and Capabilities of Table-Driven Patterns

With table-driven patterns, you have the following options:

- ☐ **Origin** - Change the origin, and pattern instances update their positions based on the origin coordinates.
- ☐ **X-Y points** - Edit the X-Y coordinates, and the pattern changes accordingly.
- ☐ **Reference point** - Change the reference, if you use a reference point other than the centroid such as a vertex, and the pattern instances update accordingly.
- ☐ **Hole Wizard** - Use table-driven patterns together with **Hole Wizard** (see **Hole Wizard** on page 4-3).
- ☐ **Edit** - Add or remove X-Y coordinates from the table. You can remove a single or multiple pattern instances. You can also delete by selecting pattern instances in the graphics area.
- ☐ **Assemblies** - Reference the feature patterns by a derived assembly component pattern to automatically insert components.
- ☐ **Performance** - Use **Geometry pattern** to speed-up performance in applicable instances.
- ☐ **Drawings** - Reference table-driven patterns in drawings.
- ☐ **Save table-driven pattern** - You can save the X-Y coordinates of a table-driven as a **.sldptab** file. You can later retrieve the **.sldptab** file, and apply the pattern of X-Y coordinates to a new part. See page 4-8.

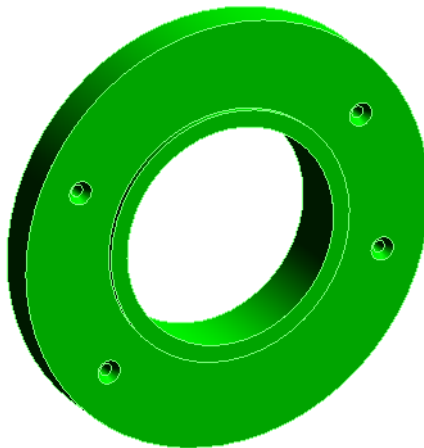
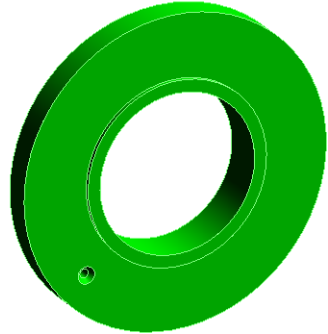
NOTE: When you enter X-Y coordinates and click **Save** in the **Table Driven Pattern** dialog box, you are *only* saving the Table Pattern (**.sldptab**). You need to apply the pattern and save the part separately.

To build a table-driven pattern:

- 1 Open **Table_driven_patrn.sldprt**.
- 2 Click **Insert, Pattern/Mirror, Table Driven Pattern**.
- 3 In the **Table Driven Pattern** dialog box, enter the following values:
 - a) Leave **Centroid** as the **Reference point**.
This specifies the center of the seed feature (**Cut-Extrude1**), as the reference point used by the X-Y values.
 - b) Select **Coordinate System1** from the FeatureManager design tree as the **Coordinate system**.
 - c) Select the **Geometry pattern** check box.
 - d) In the **Items to copy** box, select **Hole1** from the FeatureManager design tree. The X-Y coordinates of the seed feature appear as the first instance, **Point 0**.
 - e) To enter X-Y coordinates, double-click the area under **Point 0**. Enter the following values for the X and Y coordinates:

X Coordinates	Y Coordinates
-3.00in	2.00in
3.25in	1.50in
3.25in	-1.50in


- 4 Click **OK** and save the part as **Table_driven_patrn_01.sldprt**.



To edit a table-driven pattern instance:

To edit table-driven patterns, you can either use **Edit Definition**, and change the X-Y coordinates in the table, or edit from the graphics area.

To edit from the graphics area:

- 1 Open **Table_driven_patrn_01.sldprt**.
- 2 Double-click the instance of the pattern you want to edit, to display the X-Y coordinates.
- 3 Double-click the X or Y coordinate and change the value in the **Modify** box.
- 4 Click  to update the table with the new coordinates.

To edit from the FeatureManager design tree:

- 1 Right click the **TPattern1** in the FeatureManager design tree, and select **Edit Definition**.
- 2 Click the X or Y coordinates you want to change, and enter the new value.
- 3 Click **OK**.

To delete a table-driven pattern instance:

- 1 Right-click **TPattern1** and select **Edit Definition**.
- 2 In the **Table Driven Pattern** dialog box, click the number of the pattern you want to delete to highlight the X-Y coordinates.
- 3 Press **Delete**.
- 4 Click **OK**.

To save a table-driven pattern instance:

- 1 Create a part with a seed feature and create a table pattern (see page 4-7).
- 2 Click **Save** in the **Table Driven Pattern** dialog box. (If you use an existing set of X-Y coordinates which you modified, use **Save As** to rename the new pattern).
- 3 Browse to the folder where you want to save the Table Pattern, and enter a name. The **.sldptab** extension is added automatically.
- 4 Click **Save** to save the Table Pattern (**.sldptab**), and **OK** in the **Table Driven Pattern** dialog box to save the part.

To load a table-driven pattern instance:

- 1 With a part open, click **Insert, Pattern/Mirror, Table Driven Pattern**.
- 2 Click **Browse** to locate the **.sldptab** file, then click **Open**.
The file name appears in the **Read a file from** box, along with the X-Y coordinates.
- 3 Enter the appropriate **Reference point** and **Coordinate system**.
- 4 Click **OK** to apply the Table Pattern to the part.

Sketch Driven Patterns

Using sketch points within a sketch, you can specify a feature pattern. Like table-driven patterns, you can use sketch-driven patterns for holes or other feature instances. Sketch-driven patterns can resolve situations in which you have:

- ❑ An irregular pattern
- ❑ A sketch as the basis for the pattern

Definitions for Sketch-Driven Patterns

To create sketch-driven patterns, you must define the following:


- ❑ **Seed features** - Create seed features that represent the geometry you want to replicate.
- ❑ **Reference point** - Select a reference point as an origin.
 - You can use the centroid as the reference point. See **Centroid** on page 4-5.
 - You can use the sketch origin or a vertex as a reference point
 - You can create a reference point, such as a sketch point, on the seed feature.
- ❑ **Reference Sketch** - Create a sketch on a face of the original part or assembly, using sketch points to designate the pattern propagation.

Characteristics and Capabilities of Sketch-Driven Patterns

With sketch-driven patterns, you have the following options:

- ❑ **Edit** - Edit the sketch that includes the locating points. You can add or remove sketch points, and the pattern updates accordingly.
- ❑ **Delete** - Remove pattern instances by editing the sketch and deleting a point.
- ❑ **Reference point** - Change the reference point to alter the pattern propagation.

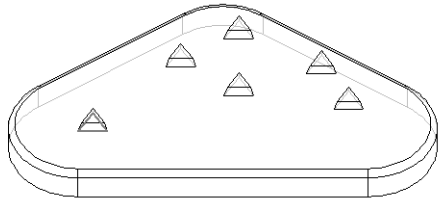
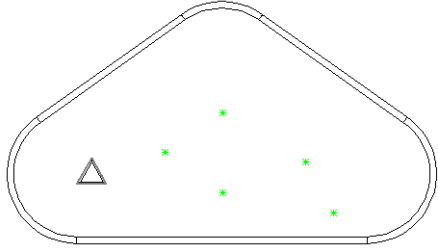
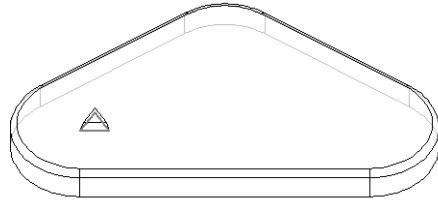
To build a sketch-driven pattern:

- 1 Open **Sketch_driven_patrn.sldprt**.
- 2 Open a sketch on the top face (with the triangular boss).
- 3 Click **Point**  or **Tools, Sketch Entity, Point** and add sketch points in an irregular pattern, approximately as shown.
- 4 Click **Insert, Pattern/Mirror, Sketch Driven Pattern**.
- 5 In the **Sketch Driven Pattern** dialog box, enter the following values from the FeatureManager design tree:
 - a) Select **Origin** in the **Reference point** box.
 - b) Select **Sketch3** in the **Reference Sketch** box.
 - c) Select **Boss-Extrude1** in the **Items to copy** box.
 - d) Select the **Geometry pattern** check box.
 - e) Click **OK**.

The sketch-driven pattern appears.

- 6 Save the part as **Sketch_driven_patrn_01.sldprt**.

NOTE: If the feature you intend to pattern includes fillets or other additions, include the features along with the seed feature in the **Items to copy** box.





To edit a sketch-driven pattern instance:

You can edit a sketch-driven pattern by:

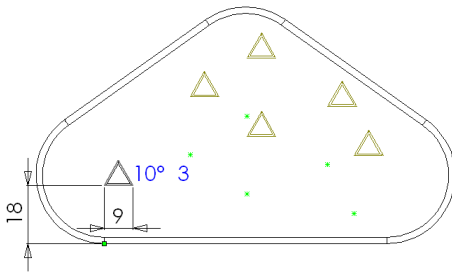
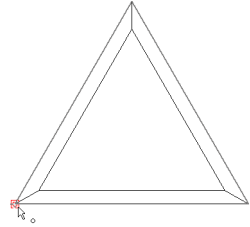
- ☐ Creating a new seed feature
- ☐ Adding or deleting the sketch that determines the pattern
- ☐ Changing the **Reference point**

To change the Reference Point:

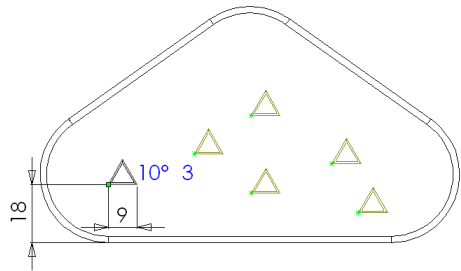
- 1 Open **Sketch_driven_patrn_01.sldprt**.
- 2 Right-click **Sketch-Pattern1** and select **Edit Definition**.
- 3 In the **Sketch Driven Pattern** dialog box, select **Point1@Origin** in the **Reference point** box and press **Delete**.
- 4 Click **Front** .
- 5 In the graphics area, select the *lower vertex* of **Boss-Extrude1**. (You may need to **Zoom to Area** .

Point6@Sketch2 appears in the **Reference point** box.

The reference points for each pattern instance, shift from using the **Origin**, to using **Point6@Sketch2** (the selected vertex) for each pattern instance of **Boss-Extrude1**.




Origin used as the Reference Point



Selected Vertex used as the Reference Point

- 6 Click **OK**.

To delete a sketch-driven pattern instance:

- 1 Right click the sketch used by the sketch-driven pattern in the FeatureManager design tree, and select **Edit Sketch**.
- 2 Select the sketch points you want to delete and press **Delete**.
- 3 Click **Rebuild**  to close the sketch and update the pattern.

Fillet Features

There are several new fillet features in the SolidWorks 2000 software. New fillet features include:

- ☐ Multiple radius fillets
- ☐ Round corner fillets
- ☐ Setback fillets


The **Fillet Feature** dialog box adjusts dynamically to reflect the type of fillet. For example, with edge-based fillets, the **Advanced Face Fillet** tab does not appear.

Multiple Radius Fillets

Using multiple radius fillets, you have the following capabilities:

- ☐ You can select a different radius value for each selected edge. You can create corners using three edges with different radii that meet at a common vertex.
- ☐ You can edit multiple radius fillets as you would any other type of fillet. Highlight the edges in the **Items to fillet** list, and the appropriate value displays in the **Radius** box.
- ☐ With multiple radius fillets, you can select surfaces as well as edges. However, you cannot assign multiple radii to faces with common edges.

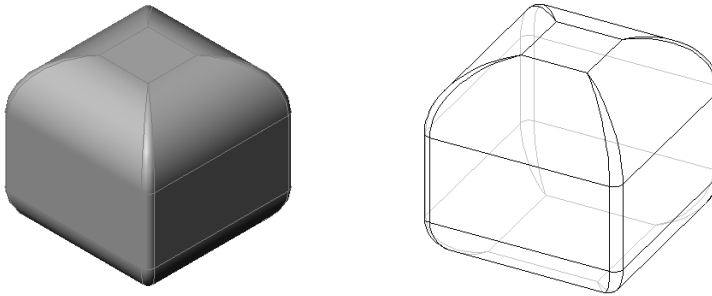
To create multiple radius fillets:

- 1 Create a cube.
- 2 Dimension where each side is 45mm.
- 3 Click **Fillet**  or **Insert, Features, Fillet/Round**. In the **Fillet Feature** dialog box:
 - a) Click the **Multiple-radius fillet** check box.
 - b) Click to clear the **Propagate to tangent faces** check box.
 - c) Set **Radius** to 10mm.
- 4 In the graphics area, select the four bottom edges of the part.

NOTE: You can enter the **Radius** first, or select the edges first.

- 5 In the graphics area, select the four *top* edges of the part
- 6 In the **Fillet Feature** dialog box, set **Radius** to 15mm.
- 7 In the **Items to fillet** box, select the remaining bottom edges (**Edge <7>**, **Edge <6>**, and **Edge <5>**), and set **Radius** to 15mm for each edge.
- 8 In the graphics area, select the four remaining (vertical) edges.
- 9 In the **Fillet Feature** dialog box, set **Radius** to 2mm.
- 10 In the **Items to fillet** box, select the remaining bottom edges (**Edge <11>**, **Edge <10>**, and **Edge <9>**), and set **Radius** to 2mm for each edge.


11 Click **OK**.

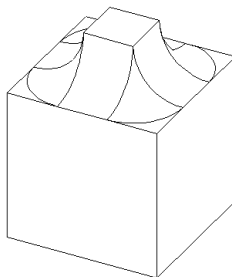
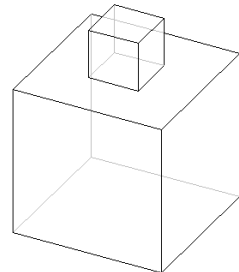


Round Corner Fillets

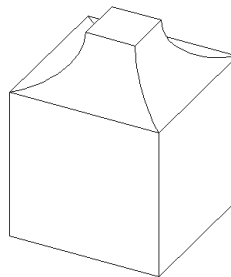
Fillets with round corners allow you to control the transition between edges at the corners. A rounded fillet blends the adjoining edges, eliminating or smoothing the sharp juncture where the two edges meet.

To create round corner fillets:

- 1 Open **Fillet_round_corners.sldprt**.
- 2 Click **Fillet**  or **Insert, Features, Fillet/Round**. In the **Fillet Feature** dialog box:
 - a) Select the **Round corners** check box.
 - b) Click to clear the **Propagate to tangent faces** check box.
- 3 In the graphics area, select the four edges at the bottom of the *smaller* cube.
- 4 Set **Radius** to 10mm.
- 5 Click **OK**.



Fillet with Round corner



Fillet without Round corner

Setback Fillets

With setback fillets, you can create a smooth transition between the blended surface, along the edge of the part, into the fillet corner. Setback fillets use a new tab, **Advanced Edge Fillet**, in the Fillet Feature dialog box.

Definitions for Setback Fillets

To create setback fillets, you must define the following in PropertyManager:


- ☐ **Transition distance** - For each edge, enter the distance to the selected vertex where the edges meet.
- ☐ **Items for setback fillet** - Select the vertex where the edges meet.
- ☐ **Edge list** - Derived from the **Items to fillet** list, plus the **Radius** in the **Fillet** tab.
- ☐ **Unassigned distance** - Select the **Set unassigned distance** check box, enter a value, and use the same distance between the vertex and the edge for all instances.

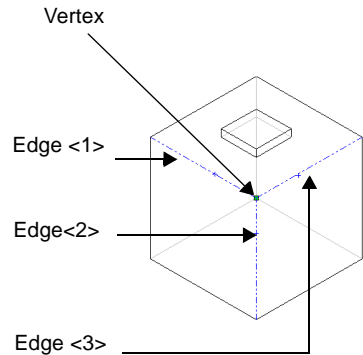
Characteristics and Capabilities of Setback Fillets

With setback fillets, you have the following options:

- ☐ **Edit** - Edit the following:
 - Select the fillet feature
 - Change the corner type
 - Change setback parameters for any fillet feature
 - Combine assigned and unassigned distances
- ☐ **Vertices** - Specify multiple vertices along a chain of blend edges for the setback fillet function.
- ☐ **Delete** - Remove one or more setback back fillet features

To create setback fillets:

- 1 Open part **Fillet_setback.sldprt**.
- 2 Click **Fillet**  or **Insert, Features, Fillet/Round**. In the **Fillet Feature** dialog box, enter the following values in the **Fillet** tab:
 - a) Select the **Multiple-radius fillet** check box.
 - b) Click to clear **Propagate to tangent faces** check box.
 - c) In the graphics area, select the three edges, as shown, for **Items to fillet**.
 - d) Set **Radius** to 5mm for **Edge <1>**.
 - e) Set **Radius** to 10mm for **Edge<2>**.
 - f) Set **Radius** to 5mm for **Edge<3>**.



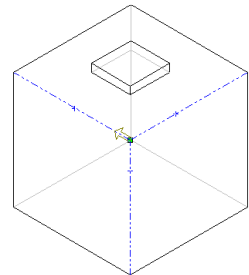
- 3 Click the **Advanced Edge Fillet** tab.
- 4 In the graphics area, select the Vertex, as shown.

Vertex<1> appears in **Items for setback fillet Vertex list**.

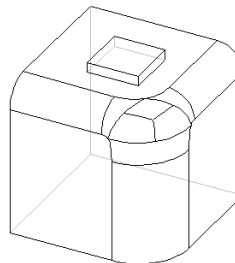
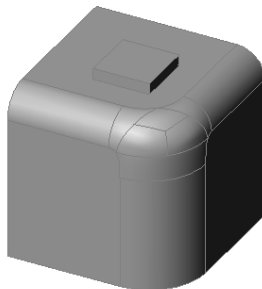
E1, **E2**, and **E3** which you previously selected to fillet, are displayed in the **Edge list**, along with a zero setback distance.

- 5 Enter the three Setback distances for **Vertex<1>**, in the **Edge list** box as follows:
 - a) Click **E1**, and set **Transition distance** to 5mm.
 - b) Click **E2**, and set **Transition distance** to 2mm.
 - c) Click **E3**, and set **Transition distance** to 2mm.

As you select an item in the **Edge list** box, an arrow appears in the graphics area indicating the direction of the fillet.



- 6 Click **OK**.



Rib Feature

Rib features in the SolidWorks 2000 application include the following enhancements.



- ☐ You can create rib features with open sketch elements. The software extends the sketches to the neighboring surfaces.
- ☐ You can create ribs from multiple, disconnected segments.

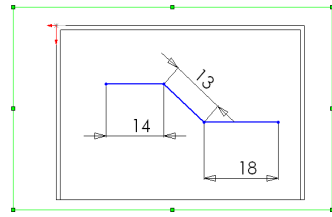
To create rib features using single or multiple, open sketch elements, you need to specify the following in the PropertyManager:

- ☐ Select an **Extrusion Direction**
 - **Normal to sketch** - follows the direction of the sketch
 - **Parallel to sketch** - extends parallel to the sketch, along the plane
- ☐ Select an **Extension Type**:
 - **Natural** - extends to follow the curve in the sketch of the rib
 - **Linear** - extends in a linear fashion from the sketch of the rib

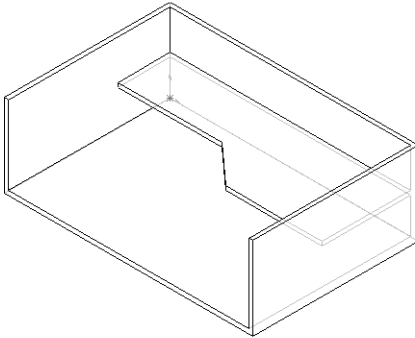
You still have the same options used with ribs for **Thickness**, **Reverse**, and **Draft Conditions**.

To create rib feature using open sketch elements:

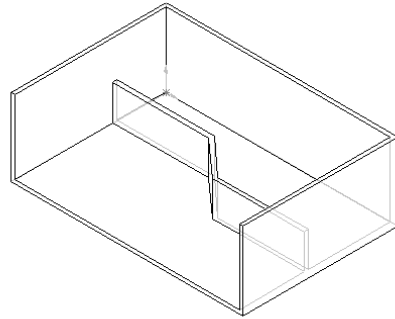
- 1 Open part **Rib_w_open_sketch.sldprt**.
- 2 Create a **Sketch**  on **Plane4**, as shown. Click **Rib** , or **Insert, Features, Rib** to display Rib feature in the PropertyManager.
- 3 In PropertyManager, set the following values:
 - a) For **Thickness**, select **Mid plane** and 1mm.
 - b) For **Extrusion Direction**, select **Parallel to sketch**.
 - c) Note the direction of the preview arrow, and click the **Flip material side** check box, if necessary.
 - d) For **Extension Type**, select **Natural**



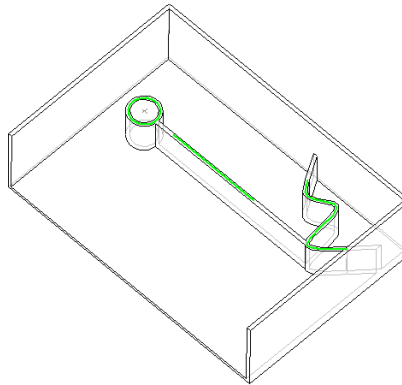
4 Click **OK**.



Extrusion Direction - Parallel to sketch



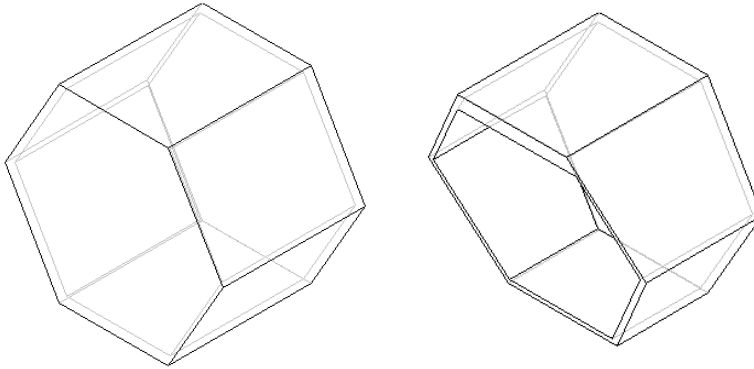
Extrusion Direction - Normal to sketch



Ribs using Multiple Disconnected Segments

Shell Feature

With the SolidWorks 2000 application, you are not required to select a face to shell a solid part. In the **Multi thickness faces** box, select **All Others** to hollow the inside of a solid part to a desired thickness. Except for the capability of leaving the **Faces to remove** box blank, the user interface remains the same.

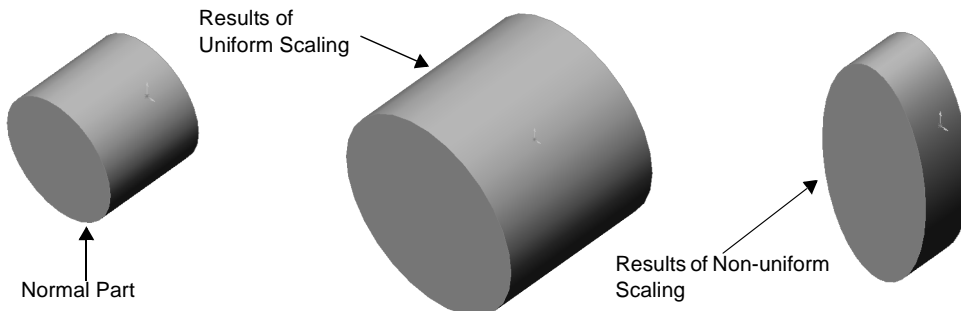


Non-Uniform Scaling

You can now select to scale using a coordinate system, in addition to the origin or the centroid. You can also select a non-uniform scaling factor by entering X-Y-Z coordinates. Non-uniform scaling also works for cavities in molds. It allows you to specify X-Y-Z coordinates, and vary the cavity in a mold.

To use non-uniform scaling:

- 1 Click **Insert, Features, Scale**.
- 2 For **Type**, select **Centroid**, **Origin**, or **Coordinate System** from the **About** list.
- 3 Click to clear the **Uniform** check box.
- 4 Enter coordinates for X-Y-Z.
- 5 Click **OK**.



Assemblies

This chapter describes general information about working with assemblies of any kind. The following enhancements are discussed:

- ☐ Display of an assembly
- ☐ Mates
- ☐ Assembly Features
- ☐ Dynamic Clearance

General Information

Suppression States

There is a new option that automatically loads any lightweight components that are out-of-date. Components that are out-of-date have a feather with stripes next to them in the FeatureManager design tree.

To resolve out-of-date lightweight components:

- 1 Click **Tools, Options**.
- 2 On the **System Options** tab, click **Performance** and make sure the **Check for out-of-date lightweight components** check box is selected.

Mass Properties

You can calculate the mass properties for a selected group of components. Previously, you could only calculate the mass properties for the entire assembly or for one selected component.

Collision Detection

You can use all types of surfaces in Collision Detection. In earlier versions, the SolidWorks software could detect collisions only on the following surface types: planar, cylindrical, conical, spherical, and toroidal. To detect collisions on all surface types, select the **Complex surfaces** check box in the PropertyManager when detecting collisions.

Creating a Component in an Assembly


When you create a component in an assembly, you must select a Document Template for the component. For more information, see **Document Templates** on page 1-5.

Move Component

You can move a component to a specified coordinate.

NOTE: You cannot break any existing mates when moving the component.

To move a component to a new location:

- 1 Select the component to move.
- 2 Click **Move Component** .
- 3 Enter an X, Y, or Z coordinate in the PropertyManager.

When you press **Enter**, the component moves to the new location.

External References

You can lock, unlock, or break the external references of an assembly component.

- When you lock the external references on a component, the existing references no longer update and you cannot add any new references to that component.
- Once you unlock the external references, you can add new references or edit the existing references.
- When you break the external references, the existing external references no longer update and you can add new references to the component.

In the FeatureManager design tree, there are new suffixes to indicate the status of locked or broken external references:

- The suffix **->*** means that the reference is locked.
- The suffix **->x** means that the reference is broken.

To lock, unlock, or break the external references:

- 1 Open **\Handle\Handle.sldasm**.

Notice that the **Handle Rod** component has an external reference. The diameter of the **Base-Extrude** feature of **Handle Rod** refers to the diameter of one of the holes in the **Connector**.

- 2 Right-click the **Handle Rod** in the FeatureManager design tree and select **List External Refs**.

The **External References For : Handle Rod** dialog box appears.

- 3 Click **Lock All** to lock all of the external references for this component.

The suffix for the component changes from **->** to **->***.

- 4 Click **Unlock All** to unlock the references for this component.

The suffix for the component changes back to **->**.

- 5 Click **Break All** to permanently break the external references for this component.



The suffix for the component changes to **->x** and the diameter of the **Base-Extrude** no longer will update if the size of the **Connector** changes.

If you want to list the broken references in the **External References For** dialog box, click the **List Broken References** check box.

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

Assembly Display

Color of Components

In **Hidden In Gray**  or **Hidden Lines Removed**  display modes, components are shown using the **Wireframe/HLR** color of the *component*. In earlier versions, the **Wireframe/HLR** color of the *assembly* was used.

To set the Wireframe/HLR color of a component:

- 1 Make sure the component that you want to change is the active document.
- 2 Click **Tools, Options**.
- 3 On the **Document Properties** tab, click **Colors** and change the **Wireframe/HLR** color. You can also select the **Apply same color to wireframe, HLR and shaded** check box to change the **Wireframe/HLR** color to the **Shaded** color.

Display Quality

You can control the quality of the **Hidden Lines Removed** and **Hidden In Gray** display modes when you have interfering bodies in an assembly. You can choose to have a more accurate display, though this can slow down performance.

To change the display quality for assemblies:

- 1 Click **Tools, Options**.
- 2 On the **System Options** tab, click **Edge Display/Selection**, then click the **High quality display of interfering bodies in HLR/HLG** check box.

Image Quality

You can control the **Shaded** quality of assembly components by changing the image quality of the assembly. When you change the image quality of the assembly, you can choose to apply the assembly's settings to each of the referenced components. In earlier releases, the quality of the component was used in the assembly, regardless of the assembly's settings.

To change the image quality for all components in an assembly:

- 1 Click **Tools, Options**.
- 2 On the **Document Properties** tab, click **Image Quality**, then click the **Apply to all referenced part documents** check box.
- 3 Move the **Shaded** slider to change the image quality. You must select **Custom** before you can move the slider.
- 4 Click **OK**, then click **Yes** when prompted to apply the changes to the components.

Assembly Mates

There are several improvements in the ways that you mate components in an assembly.

- ☐ In addition to an edge, vertex, or face, you can now select an axis or plane as a mate reference. A mate reference identifies which entity of a part is used for SmartMates when you insert a component. In earlier releases of the SolidWorks software, you could use planes and axes for SmartMates, but not for mate references.
- ☐ You can *no longer* add SmartMates when rotating a component in an assembly.
- ☐ When you add a **Distance** or **Angle** mate, the default value is the current distance or angle between the selected entities.
- ☐ You can control the alignment of a **Tangent** mate between any entities.

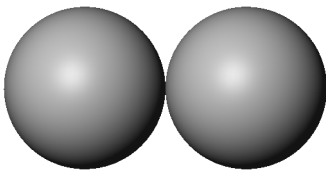
Spherical and Conical Mates

You can add mates between spherical or conical faces and other entities and control the alignment of the **Tangent**, **Coincident**, and **Distance** mates. You can add the following mates:

	Tangent	Coincident	Distance	Concentric
Sphere and Sphere	✓		✓	✓
Sphere and Plane	✓		✓	
Sphere and Cone	✓			
Sphere and Cylinder	✓		✓	
Sphere and Point			✓	✓
Sphere and Line			✓	✓
Cone and Cone		✓	✓	✓
Cone and Line				✓
Cone and Cylinder				✓

NOTE: Coincident and Distance mates between two cones must use cones of the same half-angle.

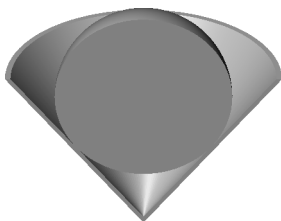
Here are some examples of the new spherical and conical mates:



Sphere and Sphere - Tangent



Sphere and Plane - Tangent



Sphere and Cone - Tangent
(Shown as a section view)




Cone and Cone - Coincident
(Shown as a section view)

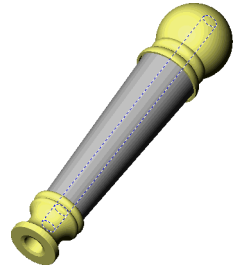
Mates of Selected Component

You can view and edit the mates of one or more selected components from the PropertyManager. You can do this whether or not the FeatureManager design tree is split into two panes.

To display the mates of a component in the PropertyManager:

- 1 Click **Tools, Options**. On the **System Options** tab, click **General**.
- 2 Make sure the **Enable PropertyManager** check box is selected.
- 3 Open **\Handle\Handle.sldasm**.
- 4 Select **Connector** and **Handle Rod** in either the FeatureManager design tree or the graphics area.
- 5 Click the **PropertyManager**  tab.

The mates for both components appear in the PropertyManager. Mates between the selected components appear in bold.



Assembly Features

Patterns of Assembly Features

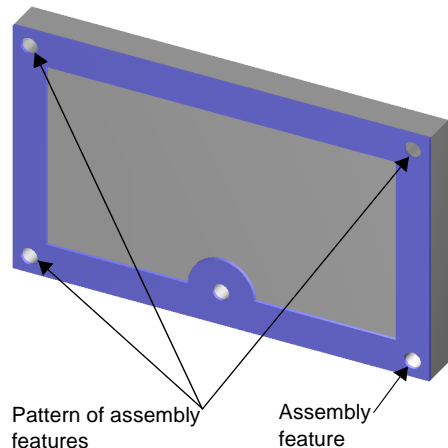
You can now create a pattern of one or more assembly features. Click **Insert, Assembly Feature**, then select the pattern type. You can create the following pattern types:

- **Linear**
- **Circular**
- **Table Driven**
- **Sketch Driven**

For more information on **Table Driven** and **Sketch Driven** patterns, see Chapter 4, “Features and Parts.”

Note the following limitations:

- You cannot pattern an assembly feature that is in a sub-assembly.
- You cannot mirror an assembly feature or a pattern of assembly features.
- You cannot create a pattern of a pattern of assembly features.



Assembly Feature Sketches

You can now have multiple closed profiles in an assembly feature sketch. In earlier versions of the SolidWorks software, you could only have one closed profile in an assembly feature sketch.



Component Patterns

You can reference a **Table Driven** or **Sketch Driven** pattern when creating a derived pattern of assembly components.

Dynamic Clearance

You can dynamically detect the clearance between components when moving or rotating a component. As you move or rotate a component, a dimension appears indicating the minimum distance between the selected components.

To dynamically detect the clearance between components:

- 1 Open **\\Slider Assembly\\Slider.sldasm**.
- 2 Click **Move Component** .
- 3 Click the **Dynamic Clearance**  check box in the PropertyManager.

NOTE: You must resolve all lightweight components to use Dynamic Clearance.


- 4 Click the box under **Dynamic Clearance**, then click any face of the red slider and the blue wall components. The names of the two components appear in the box.
- 5 Click **Resume Drag**, then drag the red slider along the yellow slide.

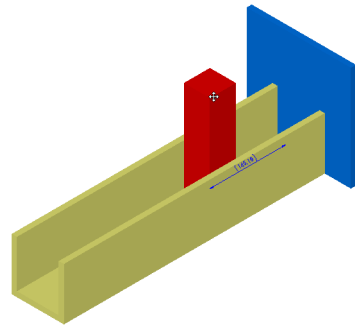
Notice a dimension appears in the graphics area and in the PropertyManager. This dimension is the minimum distance between the selected components; the dimension dynamically updates as you drag the slider.

Also in the PropertyManager, a value appears in brackets. This is the minimum distance found during the drag operation.

- 6 Drag the red slider so it is partially inside the blue wall.

Notice that the dimension disappears when the two components collide because there is no longer a minimum distance between the components.

- 7 Position the red slider where you want, then click **Move Component**  again to finish.



Drawings and Detailing

You can create 2D drawings from the 3D solid parts and assemblies you design. This chapter describes the enhancements to drawings in the following areas:

- ☐ General Information
- ☐ Drawing Views
- ☐ Dimensions
- ☐ Annotations
- ☐ RapidDraft

General Information

Templates and Sheet Formats

In SolidWorks 2000, the term **Template** is replaced in most instances by **Sheet Format**.

- **Edit Template** is **Edit Sheet Format**.
- **Save Template** is **Save Sheet Format**.
- When opening a new document, you open a *template*.
- When you add a new sheet to a drawing document, you specify a *sheet format*.
- When you edit a sheet format and save it to file, you are saving a sheet format.
- Drawing template files have the extension **.drwdot**.
- Sheet format files have the extension **.slddrt**.


You can specify the location of sheet formats in the **Options** dialog box. On the **System Options** tab, click **File Locations**. In the **Show folders for** section, select **Sheet Format**.

You can add customized templates for drawings to the SolidWorks system, and you can add tabs to the **New SolidWorks Document** dialog box. See **Document Templates** on page 1-5 for more information.

Drawing Name

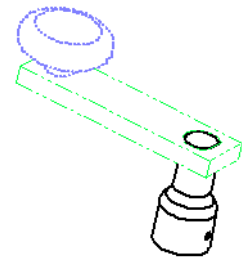
A new drawing takes the name of the first model inserted. The name appears in the title bar. When you save the drawing, the name of the model appears in the **Save As** dialog box as the default file name with the extension **.slddrw**. You can edit the name before saving the drawing.

Hide/Show Dimensions Tool


A new tool, **Hide/Show Dimensions** , has been added to the Drawing toolbar. The functionality is available in both SolidWorks 99 and SolidWorks 2000 under **View, Hide/Show Dimensions**.

Layers

You can now add components to layers, in either part or assembly drawings. The **Component Display Properties** dialog box includes a list for selecting a named layer for the component. The dialog box controls line style and weight. However, if you set **Line Style** and **Line Weight** to **Default**, the settings for the layers are used. The illustration at right shows the effect of setting each part in a different layer, with different colors, line styles, and line thicknesses set in the **Layers** dialog box.



To access the **Component Line Font** dialog box, right-click a component, either in the graphics area or in the FeatureManager design tree and select **Component Line Font**.

The new Layer toolbar includes a window that displays the current layer and the **Layer Properties** tool . You can move an entity into a layer by selecting it and choosing a layer from the list. With the **Layer Properties** tool, you can create new layers, set the layer **Color**, **Style**, and **Thickness** properties, and move components into a layer.

Drawing Views

Auxiliary View

You can now select a silhouette edge, an axis, or a sketched line as the reference entity to create an **Auxiliary** view. In previous releases, only an edge could be the reference.

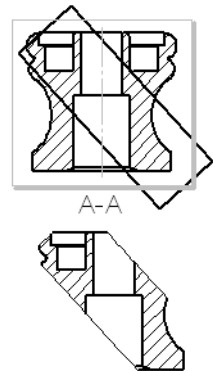
While creating an **Auxiliary** view, you can toggle the view orientation (front or back) by pressing **Tab** while dragging the view. After you have placed the view, you can toggle the view orientation by double-clicking the arrow.

Detail View


You can change the position and size of a detail profile. The detail view changes as the profile changes.

If you use a shape other than a circle as the profile for a Detail View, in the parent view you can now choose to display either the profile, as shown at right, or its circle. To display the profile, click **Profile** in the **Detail Circle Properties** dialog box.


If you select **Circle** as the display in the **Detail Circle Properties** dialog box, now you can select the **Style** of the detail note: **Default**, **Broken Circle**, **Beside Profile**, **With Leader**, or **No Leader**. You can drag the note anywhere except inside the profile.



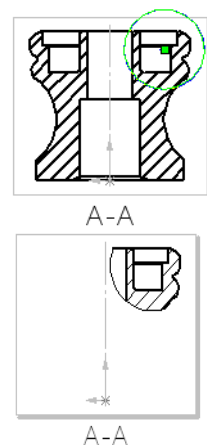
Crop View

A new tool, **Crop View** , has been added to the Drawing toolbar. You can crop any drawing view except a Detail View. This approach can save steps because you do not create a new view. For example, instead of creating a Section View and then a Detail View, then hiding the unnecessary Section View, you can crop the Section View directly.

To crop a view:


- 1 Activate an existing view.
- 2 Draw a closed profile with a sketch tool such as a circle.
- 3 Click **Crop View** , or **Tools, Crop View, Crop**.

The view outside the profile disappears.




To edit the cropped view, select the view and click **Tools, Crop View, Edit Crop**. To remove the cropping, select the view and click **Tools, Crop View, Remove Crop**.

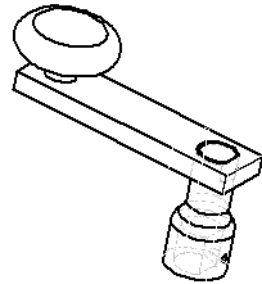
Show/Hide Tool

A new tool, **Show/Hide** , has been added to the Drawing toolbar. In earlier releases, hidden lines were either all shown or all hidden. If they were all shown, individual lines could be hidden. The **Show/Hide** tool shows or hides all the hidden lines of selected features and components. You can also access this functionality through the **Drawing View Properties** dialog box, which now has a **Show Hidden Edges** tab.

To show the hidden lines of a feature or component:

- 1 Click **Show/Hide**  on the Drawing toolbar. The **Show Hidden Edges** dialog box appears.
- 2 Click a feature or component in a drawing view or the FeatureManager design tree to add it to the list of items to be shown with hidden lines in Hidden Lines Removed mode.
- 3 Click **OK**.

The illustration at the right shows how you can display the hidden lines of a selected part (the crank shaft) while leaving the hidden lines of the other parts (crank arm and knob) invisible.



Hide/Show Components

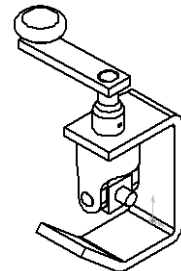
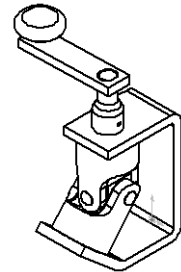
A new tab, **Hide/Show Components**, in the **Drawing View Properties** dialog box allows you to hide or show components in an assembly drawing. In the illustration, the complete assembly is shown first, then a yoke is hidden.

To hide or show components:

- 1 Right-click in a drawing view and select **Properties**. The **Drawing View Properties** dialog box appears.
- 2 Select the **Hide/Show Components** tab.
- 3 Select components in the drawing to add to the list of hidden components.

To delete components from the list, select the item in the list and press **Delete**.

- 4 Click **OK**.



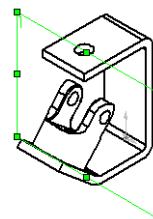
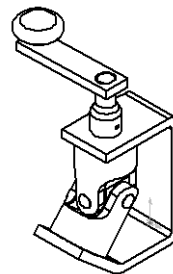
Hide Components Behind Plane

You can hide components of an assembly behind a plane. This procedure is a quick selection method for **Hide/Show**, described in the previous section.

Only those components that are completely behind the plane are hidden.

You can choose the direction to hide and the distance from the plane. The X, Y, and Z coordinates of the plane distance are shown in the dialog box. In the illustration, the complete assembly is shown first, then the plane hides the crank assembly, one yoke, the spider, and pins.

Components can be on only one list at a time. For example, if a component is on the **Hidden Behind Plane** list, the component is removed from this list when it is added to the **Hide/Show Components** list.



To hide components behind a plane:

- 1 In the FeatureManager design tree, select a plane.
- 2 Right-click and select **Hide Behind Plane**.
The **Hide Behind** dialog box appears.
- 3 Change the distance of the plane in the **Distance** box to move the plane, if necessary.
The X, Y, Z coordinates of the plane appear in the **Plane point** section.
- 4 Select the **Reverse direction** check box, if necessary.
- 5 Click **OK**.

To show the components again:

- 1 Right-click the view, and select **Properties**.
The **Drawing View Properties** dialog box appears.
- 2 Select the **Hide/Show** tab.
- 3 Select all the items in the list that are behind the plane and press **Delete**.
- 4 Click **OK**.


Dimensions

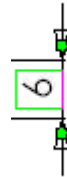
Importing Dimensions

When you import dimensions into a drawing, the following rules now apply:

- The dimensions are imported only once for a part, even if the part shows in multiple instances in an assembly.
- Dimensions that have the same value and that could overlap or be parallel are not imported. This affects linear dimensions only.
- Dimensions of draft features are now imported.

Dimension Arrows

Circular handles now appear on dimension arrows when dimensions are selected, as shown at right. When the pointer is over the circular handles, it changes to . When you click on an arrowhead handle (on *either* handle if there are two for the dimension), the arrows flip outside or inside. When you *right-click* on a handle, a list of arrowhead styles appears. You can change the style of any dimension arrowhead *individually* by this method.



A second list of arrow styles now appears in the **Dimension Properties** dialog box. You can choose separate styles for each arrow when there are two arrows for a dimension. This feature supports the JIS dimensioning standard.

Two arrowhead style menus appear on the PropertyManager tab when you select the JIS standard. Only one arrowhead style menu appears for other dimensioning standards such as ANSI and ISO. The default arrowhead style reflects the style for the selected standard.

JIS standard



ISO standard

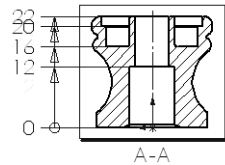


Display Options

The shortcut menu for dimensions now includes a **Display Options** menu item. This shortcut menu contains dimension items such as **Offset Text**, **Center Text**, **Jog**, and so on.

A new display option for linear dimensions and for radial dimensions displayed as linear is **Add Slant**. **Add Slant** displays slant handles on the witness line endpoints. **Add Slant** changes to **Remove Slant** in the shortcut menu for dimensions that already have slant.

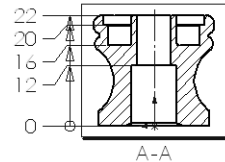
A new display option for ordinate dimensions is **Re-Jog Ordinate**, which applies the automatic jogging algorithm to the selected ordinate dimensions. For example, if you remove the jog from several ordinate dimensions, you may then decide to return to the automatic jogging.




To re-jog ordinate dimensions:

- 1 Select all the ordinate dimensions to be repositioned.
- 2 Right-click and select **Display Options, Re-Jog Ordinate**.

The illustration at right shows ordinate dimensions that have been straightened and then re-jogged.



Dimension Properties Dialog Box

A degree symbol button  is now available in the **Modify Text** section of the dimension **Properties** dialog box.

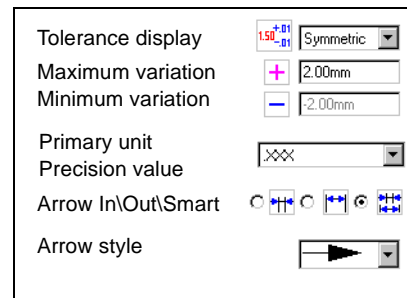
Precision

In SolidWorks 99, you could access the precision of dimensions and tolerances in the **Dimension Properties** dialog box. It was also possible to edit multiple dimensions at one time. In SolidWorks 2000, this capability has been extended to the PropertyManager tab.

To activate the PropertyManager for precision:

- 1 Click **Tools, Options**.
- 2 On the **System Options** tab, click **General**, select the **Enable PropertyManager** check box.
- 3 Click **OK**.
- 4 In the document, select one or more dimensions.

The tab changes automatically from the FeatureManager design tree to the PropertyManager.



If the precision is not set to be the document default, the tolerance precision changes its value according to what you type. For example, if you type 1.000 for the tolerance value, its precision becomes 3 digits.

Gap Distance with Silhouettes

The gap distance between a witness line and the model entity is now more consistent. The gap distance on a dimension is based on a vertex. Regular edges have vertices that you can select to create dimensions. SolidWorks 2000 now allows you to select silhouette vertices so that dimensions have a consistent appearance.

Section Line Drag Handles

Drag handles have been added to section lines so you can manipulate the text and segments of the line to meeting drafting standards, particularly JIS. You can drag:

- Text
- Leader
- Section line length

Annotations

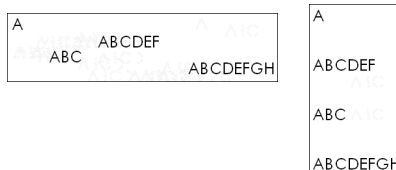
Multiple Annotations

In earlier releases of SolidWorks, you could create only one instance of an annotation at a time. In SolidWorks 2000, you can create multiple Notes, Surface Finish Symbols, Datum Feature Symbols, Datum Targets, and Geometric Tolerance Symbols without closing the dialog box. The procedure described in the section on **Notes** on page 6-9 applies to the functionality of the other types of annotations.

Align Annotations

You can align annotations. The new annotation alignment applies to Notes, Geometric Tolerance Symbols, Datum Feature Symbols, Datum Targets, Surface Finish Symbols, and Weld Symbols. Alignment is accessible three ways:

- Click **Tools, Align Annotations**.
- Select the annotations, then right-click and select **Align**.
- Display the new Align toolbar (shown at right), then select the annotations and click an alignment tool. The alignment tools are **Align Left**, **Align Right**, **Align Top**, **Align Bottom**, **Space Across**, and **Space Down**. In the example below, the original at left is aligned left and spaced down, with the result on the right.



Annotations snap to the grid and also display inferencing lines for alignment vertically and horizontally.

Annotation Arrows


You can change the arrowhead style on annotations (Notes, Surface Finish Symbols, Geometric Tolerances, Datum Targets, Welds) by right-clicking the arrow to access a list of styles.

Notes

In earlier releases, you started a note by selecting its position first. In SolidWorks 2000, additional functionality includes:

- Starting a note without selecting its position first.
- Placing multiple notes without closing the dialog box.
- Editing existing note text in place by double-clicking the note. (You can still edit the note text in the **Properties** dialog box.)

To start a new note:

- 1 With nothing selected, click **Note** . The **Properties** dialog box appears.
- 2 Type text in the **Note text** box and select options as desired.
- 3 Place the note as follows:

With the dialog box still open, click in the graphics area to place the note. Click as many times as you like to place multiple copies of the note.

- 4 Click **OK** to close the dialog box.

Other new functionality with notes (and other annotations) includes:

- **Placement.** When you select **Always show leaders**, your first click places the leader, and the second click places the note.
- **Multiple notes.** When you place multiple notes, you can change the text and other items in the dialog box between notes.
- **Dragging.** While dragging the note and before placing it, press **Ctrl**. The note stops moving, but the leader continues, lengthening the leader. While still pressing **Ctrl**, click to place the leader. Click as many times as you like to place additional leaders. Release **Ctrl** and click to place the note.


Geometric Tolerance

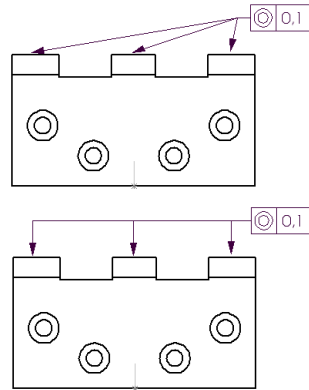
Geometric Tolerance functionality is extended:

- Placement of the symbol, with or without leaders, anywhere in a drawing, part, assembly, or sketch, and attached to anywhere on a dimension line
- Multiple 90-degree leaders

In SolidWorks 99, multiple fan leaders are available. SolidWorks 2000 adds the option of multiple 90-degree leaders. The two types of leaders are shown at right.

To display multiple 90-degree leaders:

- 1 Click **Geometric Tolerance** .
- The **Properties** dialog box appears.
- 2 Select symbols, type in tolerances, and so on.
- 3 Click **Options**.
- 4 Select **Always show leaders**.
- 5 Select the **Perpendicular leaders** check box.
- 6 Click **OK** to close the **Geometric Tolerance Options** dialog box.
- 7 Click in the graphics area to place the leader.
- 8 Click again in the graphics area to place the symbol.
- 9 Click **OK** to close the **Properties** dialog box.
- 10 Select the symbol.
- Handles appear on the symbol and leader.
- 11 Press **Ctrl** and drag the leader handle to create another leader.



Center Mark

Center marks can now be added to silhouette edges.

Cosmetic Threads

The cosmetic thread dialog box retains settings, thus facilitating inserting multiple cosmetic threads. You can also insert multiple threads by selecting multiple edges, interactively selecting or deselecting edges.

Conical cosmetic threads are now available. Instead of **Major Diameter** or **Minor Diameter**, you specify **Conical Offset**. If the conical thread does not end at a flat face, it is trimmed by the curved face.

Secondary Weld Fillet

You can now add secondary weld fillet information in the **Weld Symbol** dialog box for standards that support secondary fillets.

Balloons

You can attach balloons to sketch entities. This functionality is useful when sketches (with no extrusions) are combined to form an assembly and then taken into a drawing.


The default balloon size has changed from **Tight Fit** to **2 Characters** so that balloons are consistent in size.

You can change the **Item Number** in a balloon, and the item number in the Bill of Materials also changes.


To change an item number in a balloon:

- 1 Double-click the balloon.
An edit box with the **Item Number** appears.
- 2 Edit the number
- 3 Click outside the box to close it.

Stacked Balloons

A new tool, **Stacked Balloon** , stacks balloons with only one leader for the set. You can stack the balloons vertically or horizontally and up or down, or left or right.

To insert stacked balloons:

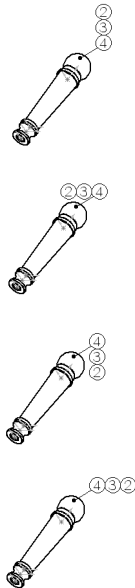
- 1 Click **Stacked Balloon**  on the Annotation toolbar or click **Insert, Annotations, Stacked Balloon**.
- 2 Select the part for the balloon with a leader.
- 3 Continue to select parts.
A balloon is added for each part selected.

Each balloon is a separate note that you can select to delete or edit.

To change the direction of the stacked balloons:

- 1 Right-click any balloon in the stack and select **Properties**.
The **Properties** dialog box appears.
- 2 In the **Angle** section, select **Up**, **Left**, **Down**, or **Right**.
- 3 Click **OK**.

You can add balloons to a stack later. Right-click one of the stacked balloons and select **Add to Stack**.



Bill of Materials

In SolidWorks 99, if rows have the same value in the Part Number column, they are combined into one row. In SolidWorks 2000, the rows are combined only when *all* fields (except for Item Number) are the same.

As mentioned in the previous section, you can edit **Item Number** in a balloon and the corresponding item number in the Bill of Materials changes.

RapidDraft Drawings

RapidDraft™ drawings have a format designed so you can open and work in drawing files without the model files being loaded into memory.

View borders in RapidDraft drawings are blue. When the referenced model is loaded into memory, the view borders change to gray or green.

If the part or assembly is needed for an operation within a RapidDraft drawing, you are prompted to load the model file. You can load the model manually by right-clicking a view and selecting **Load Model**.

Advantages of RapidDraft Drawings

You can send RapidDraft drawings to other SolidWorks users without sending the model files. You also have more control over updating the drawing to the model. Members of the design team can work independently on the drawing, adding details and annotations, while other members edit the model. When the drawing and the model are synchronized, all the details and dimensions added to the drawing update to any geometric or topological changes in the model.


Performance

The time required to open a drawing in RapidDraft format is significantly reduced because the model files are not loaded. Because the model data is not stored in memory, more memory is available to process drawing data, which has significant performance implications for large assembly drawings. You have control over when to load the model, which takes time to load and update the drawing.

File Size

The RapidDraft format requires storing more edge data and less surface data. Therefore, some files are larger when converted to RapidDraft, while others are smaller. In general, if your drawings have section views, the file size should decrease. If the drawings do not have section views, the file size *may* increase. File size is directly related to the number of visible edges in the drawing. For example, if your parts have patterns of features with many instances, it is more likely that the file size will increase when converted to RapidDraft format.

View Updates

Some changes, such as changes to a section line or a detail circle, require a view update. When a drawing view requires an update, the view is displayed with a gray crosshatch pattern. Click **Rebuild**  to update the view.

New Drawings and Existing Drawings

When starting a new drawing, you can choose to create the drawing as a RapidDraft drawing. When you open a new drawing document, the check box **Create RapidDraft Drawing** appears in the **New SolidWorks Document** dialog box.

When you open an existing drawing, SolidWorks recognizes whether the drawing is a RapidDraft drawing. If the drawing you select is not a RapidDraft drawing, the check box **Convert to RapidDraft** appears. If you convert and then save the drawing, the next time you open it, SolidWorks recognizes the drawing as a RapidDraft document and does not display the conversion check box.

When you convert a drawing to RapidDraft, the drawing and model are both loaded in memory. When the conversion is complete, save the drawing. Close the drawing, which also closes the model, then open the drawing again. The drawing is now in RapidDraft mode.

NOTE: Once a drawing is converted to RapidDraft format, it cannot be converted back. You are advised to save a version of the unconverted file.

Some SolidWorks 95 and 96 drawings require the model to be loaded even after conversion to RapidDraft.

Operations Available in RapidDraft Drawings Without the Model

Following are some of the capabilities available when the model is not loaded:

- ☐ Save and Open drawings
- ☐ Add
 - Dimensions
 - Annotations (Note, Surface Finish, Datum Feature Symbol, Datum Target, Geometric Tolerance, Weld Symbol, Center Mark)
 - Empty views
 - Balloons (with asterisks, which are resolved to item numbers when the model is loaded)
- ☐ Change
 - Scales of sheets or views
 - Line Color, Line Style, and Line Thickness of model edges and sketch entities
 - View alignments

- ☐ Select
 - Imported planes, sketches, origins, and axes
 - Section view edges
 - Cosmetic threads
 - Model edges and vertices
- ☐ Inference, reference, and dimension to model edges, vertices, and center points
- ☐ Hide edges
- ☐ Update the drawing with the Rebuild tool

Operations Requiring the Model

When the model is not loaded, the model entities do not appear in the FeatureManager design tree and the following operations are not available:

- ☐ Import
 - Design Tables
 - Model items (dimensions, annotations, origins, planes, axes, or sketches)
- ☐ Add
 - New views (Standard, Named, Projection, Section, Detail, Relative, Auxiliary, Broken, Cropped)
 - Bills of Materials
 - Hole Callouts or Cosmetic Threads
- ☐ Change
 - Values or properties of feature dimensions imported from the model
 - Hatch scales
 - Cropped View
- ☐ Select
 - Model faces
 - Surfaces
 - Component Line Font item from shortcut menu
- ☐ Cut and Paste views
- ☐ Hide/Show Components or Hide Behind Plane

Index

3D sketching
splines 2-11

A

advanced face fillet 4-12
align annotations 6-8
annotations 1-7, 6-8–6-11
align 6-8
geometric tolerance 6-9
multiple 6-8
notes 6-9
weld symbol 6-10
arrows
handles 6-6
assembly
component color 5-4
components on layers 1-13
conical mates 5-5
create component 5-2
features 5-7
lightweight 5-2
move component 5-2
pattern a component 5-8
pattern a feature 5-7
spherical mates 5-5
auxiliary view 6-3

B

balloons 6-10
item number 6-11
stacked 6-11
base part 4-2
bill of materials 6-11
break references 5-3

C

center mark 6-10
centroid 4-5, 4-7, 4-9
check entity 4-2
circular sketch step and repeat 2-5
Collision Detection 5-2
color
gradient background 1-11
part color in assemblies 5-4
schemes 1-11
component line font 6-2
configuration-controlled data 1-14
conical cosmetic threads 6-10
conical mates 5-5
construction geometry 2-2
Conversion Wizard x
coordinate system 4-3, 4-5, 4-18
Copy Options Wizard xi
cosmetic threads 6-10
counterbore 4-4
countersink 4-4

Index

crop view 6-3
curvature evaluator 2-8

D
degree symbol 6-7
delete faces. *See* surface bodies 3-13
derived parts 4-2
detail view 6-3
dimensions 6-6–6-8
 arrows 6-6
 degree symbol 6-7
 display options 6-6
 gap distance 6-8
 importing to drawings 6-6
 link values 4-2
 precision 6-7
 re-jog ordinate 6-7
 slant 6-6
display options 6-6
display quality 5-4
document
 references 1-12
 where used 1-12
document options. *See* options
documents
 opening new 1-5
 view 1-6
drawings 6-1–6-11
 auxiliary view 6-3
 crop view 6-3
 detail view 6-3
 importing dimensions 6-6
 names 6-2
 RapidDraft 6-12
 show hidden lines 6-4
DXF/DWG files 1-13
Dynamic Clearance 5-8
dynamic transparency. *See* performance

E
edit
 circular sketch patterns 2-6
 linear sketch patterns 2-7
 polygons 2-3
end condition 3-3
entity check 4-2
equations 4-2

export
 DXF/DWG files 1-13
 IGES files 1-13
 STEP files 1-14
extend edge, face. *See* surface bodies
extension type 3-3
external references 5-3

F
Fast HLR. *See* performance
fillets 4-12–4-15
 face blend 3-11
 multiple radius 3-11, 4-12
 round corner 4-13
 setback fillets 4-14–4-15
filter surface bodies 3-2

G
gap distance 6-8
geometric tolerance 6-9
Getting Started ix, xi
gradient color 1-11

H
handles
 arrow 6-6
 section line 6-8
hide
 components 6-4, 6-5
 dimensions 6-2
 hidden lines 6-4
HLR display shaded 1-4
hole definition *see* hole wizard
hole wizard
 legacy 4-4
 table-driven patterns 4-6
hyperlinks 1-12

I
IGES files 1-13
image quality 5-4
import
 DXF/DWG files 1-13
 IGES files 1-13
 options 1-12
 Parasolid files 1-13
 sketch entities 1-13

inspect curvature, spline 2-8
intersection curve 2-4

J

JIS standard 6-6, 6-8

K

knit surfaces. *See* surface bodies 3-7

L

layers

 component line font 6-2

leaders

 notes 6-9

 perpendicular 6-10

lightweight components 5-2

line font 6-2

linear sketch step and repeat 2-7

link values 4-2

lock references 5-3

M

mass properties 4-3, 5-2

middle mouse button functions 1-9

mirror part 4-2

moving frame

 PropertyManager display 2-10

 spline 2-9

multiple

 annotations 6-8

 leaders 6-9

mutual trim 3-7

N

names

 drawings 6-2

non-uniform scaling. *See* scaling

notes 6-9

O

options

 access 1-10

 document options 1-9

 IGES 1-13

 import 1-12

 system options 1-9

 transparency quality 1-15

out-of-date components 5-2

P

parametric sketch patterns 2-5

Parasolid files 1-13

pattern

 circular step sketch and repeat 2-5

 linear step sketch and repeat 2-7

performance 1-4

 dynamic transparency

 Fast HLR

 RapidDraft drawings 6-12

perpendicular leaders 6-10

pipe tap 4-4

planes

 hide components behind 6-5

 trim tools

pointer with surface bodies. *See* surface bodies

polygon 2-2

precision 6-7

profile

 crop view 6-3

 detail view 6-3

project curve on surface 3-13

PropertyManager 1-8, 6-6, 6-7

Q

quality

 display 5-4

 image 5-4

R

RapidDraft drawings 6-12

reference geometry. *See* surface bodies

re-jog ordinate 6-7

ribs 4-16–4-17

 multiple open sketch elements 4-16

 open sketch elements 4-16

S

scaling 4-18

section line drag handles 6-8

service packs xi

sheet format 6-2

shell 4-18

shortcut menus 1-8

show

 components 6-4, 6-5

 dimensions 6-2

 hidden lines 6-4

silhouettes 6-8, 6-10

Index

- sketch
 - circular sketch pattern 2-5
 - linear sketch pattern 2-7
 - sketch entities
 - attaching balloons 6-10
 - sketch entity
 - polygon 2-2
 - spline 2-8
 - text tool 2-2
 - sketch from drawing 1-13
 - sketch patterns, parametric and editable 2-5
 - sketch-driven patterns 4-9–4-11
 - changing reference point 4-11
 - creating 4-10
 - deleting 4-11
 - features 4-9
 - slant 6-6
 - SolidWorks Explorer 1-11
 - spherical mates 5-5
 - spline
 - 3D sketch 2-11
 - add relations 2-8, 2-10
 - curvature evaluator 2-8
 - inspect curvature 2-8
 - moving frame 2-9
 - split display 1-7
 - split line. *See* surface bodies
 - stacked balloons 6-11
 - STEP files 1-14
 - surface bodies 3-2
 - creating extensions 3-4–3-6
 - delete faces 3-13
 - extend edge, face 3-3
 - face blend fillet 3-12
 - hide and show 3-14
 - knit surfaces 3-7, 3-13
 - multi radius fillet 3-11
 - pointer 3-2
 - split line 3-13
 - trim 3-7
 - surfaces. *See* surface bodies
 - system options. *See* options
- T**
- table-driven patterns 4-5–4-8
 - creating 4-7
 - editing 4-8
 - features 4-5
 - tabs 1-6
 - uses
 - tap 4-4
 - templates
 - creating 1-5
 - drawing 6-2
 - file types 1-5
 - templates. *See also* tabs 1-6
 - text tool, sketch entity 2-2
 - toolbars
 - align 1-2
 - annotation 1-2
 - curves 1-2
 - drawing 1-3
 - mold tools 1-3
 - selection filter 1-3
 - sheet metal 1-3
 - sketch tools 1-3
 - standard 1-3
 - surfaces 1-4
 - view 1-4
 - trim surface. *See* surface bodies
 - trim tools
 - planes 3-7
 - sketch extrusion 3-7
 - surface 3-7
- U**
- updates, SolidWorks service packs xi
- X**
- x-y coordinates 4-5, 4-7



SolidWorks 2000 Document Order Form

Getting Started Manual

SolidWorks will ship the SolidWorks 2000 Getting Started manual, as ordered on this form, to registered customers who have upgraded or are on the SolidWorks Subscription Service plan.

Orders can be processed only if *all* information is received. Type or print clearly.

Quantity required _____

Serial Numbers _____

Include a serial number for each set requested. (For example: 0000 0000 1234 1234)
Attach additional sheet if necessary.

Language required:

- | | |
|----------------------------------|----------------------------------|
| <input type="checkbox"/> English | <input type="checkbox"/> Italian |
| <input type="checkbox"/> French | <input type="checkbox"/> Russian |
| <input type="checkbox"/> German | <input type="checkbox"/> Spanish |

Mail your request to:

SolidWorks Corporation
Attn: Books
300 Baker Avenue
Concord, MA 01742 USA

Fax your request to:

+1-978-371-4996

Order online:

<http://www.solidworks.com/docorders>

Please send my package to:

Company _____

Attention _____

Address _____

_____ **Country** _____

Telephone Number _____