



SolidWorks[®] 99

What's New

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U.S. Patent 5,815,154

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About This Book

There are many new features available in SolidWorks® 99. To help you learn these new features, this book has been created to highlight the new functionality. This manual is for experienced users and assumes you have a good working knowledge of SolidWorks 98Plus.

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 99 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** directory.

TIP: To install the *User's Guide* and *Tutorial* 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 document 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 99** in the online help.

Moving to SolidWorks 99

Converting Older SolidWorks Files to SolidWorks 99

Because of changes to the SolidWorks files with the development of SolidWorks 99, opening a SolidWorks document from an earlier release may take more time than you are used to experiencing. However, once the file has been opened and saved, subsequent opening time 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 99 format. Depending on how many files you have, the conversion process may take a while, but once it is done, you will not experience the slow initial opening of your files.

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

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

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

Copy Options Wizard

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

To use the Copy Options Wizard:

- 1 From the **Programs** list, select **SolidWorks 99, Copy Options Wizard**.
- 2 To capture your SolidWorks options information from the registry, click **An existing user** and select your name from the 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 99 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.

User's Guide and Tutorial

The *User's Guide* and *Tutorial* are 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 the Microsoft **Start** button, select **Programs**, and select the **SolidWorks 99** program group. Click **User's Guide** or **Tutorial**.

SolidWorks Fundamentals



















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

- Toolbars
- Printing parts and assemblies
- Reference geometry
- Import and export

General Information

Toolbars and Tools

There are two new toolbars, Reference Geometry and Tools. The tools on the Reference Geometry toolbar used to be on the Features toolbar. There are also many new tools, as shown in the following table, which are explained in subsequent chapters.


Icon	Tooltip	Equivalent Menu Selection
Standard toolbar		
	Edit Color	None
Sketch toolbar		
	3D Sketch	Insert, 3D Sketch
	No Solve Move	Tools, Sketch Tools, No Solve Move
Sketch Tools toolbar		
	Split Curve	Tools, Sketch Tools, Split Curve
	Linear Sketch Step and Repeat	Tools, Sketch Tools, Linear Step and Repeat
	Circular Sketch Step and Repeat	Tools, Sketch Tools, Circular Step and Repeat
	Parallelogram	Tools, Sketch Entity, Parallelogram
Assembly toolbar		
	Smart Mates	None
Tools toolbar		
	Measure	Tools, Measure
	Equations	Tools, Equations
	Mass Properties	Tools, Mass Properties
Line Format toolbar		
	Layer properties	None
Features toolbar		
	Linear Pattern	Insert, Pattern/Mirror, Linear Pattern
	Circular Pattern	Insert, Pattern/Mirror, Circular Pattern
	Mirror Feature	Insert, Pattern/Mirror, Mirror Feature
	Sweep	Insert, Base, Sweep or Insert, Boss, Sweep
	Loft	Insert, Base, Loft or Insert, Boss, Loft
	Insert Rip	Insert, Features, Sheet Metal, Rip

Macros as Toolbar Buttons

You can assign your own macros to toolbar buttons.

To assign a user-defined macro to a toolbar button:

- 1 Click **Tools, Customize, Macros**.
- 2 Click **Add**.
 - a) In the **Path** box, type the path of the macro, or browse to the desired file.
 - b) In the **Name** box, type the name for the macro. If you do not type a name, the macro filename is used.

The name is used as the tooltip for the toolbar button.
 - c) Click **Apply** to add the macro.
- 3 Click the **Commands** tab.
- 4 Select **Macro** from the **Categories** list.
- 5 Drag **User Macro 1**  to any toolbar, then click **OK**.

You can now start your macro by clicking the toolbar button.

NOTE: The **User Macro** buttons may already appear on your Macro toolbar, if that toolbar is visible.

Auto Activate Document Toolbars

A new check box, **Auto activate document toolbars**, has been added to the **Toolbars** tab of the **Customize** dialog box. When selected, the SolidWorks software activates the toolbars appropriate to the active document type. When cleared, the visible toolbars remain constant regardless of the active document type. For example, if you hide the Drawing toolbar and you switch to a drawing document:

- If the **Auto activate document toolbars** check box is selected, the Drawing toolbar becomes visible.
- If the **Auto activate document toolbars** check box is cleared, the Drawing toolbar remains hidden.

Scaling of OLE Objects

You can now change the size or scale of an OLE object in a SolidWorks document.

To change the size or scale of an OLE object:

- 1 Right-click an OLE object, and select **Properties**.

The **OLE Object Property** dialog box appears.
- 2 Type a value in either the **Width**, the **Height**, or the **Scale** box.

When you change any one of the values, the others update automatically so that the proportions of the object are maintained.
- 3 Click **OK** to close the dialog box and apply the new values.


Open Only SolidWorks Documents

You can now group SolidWorks parts, assemblies, and drawings in the **Open** dialog box.

To open a SolidWorks document of any type:

- 1 Click **File, Open**.
The **Open** dialog box appears.
- 2 In the **Files of type** list, click **SolidWorks Files (*.sldprt; *.sldasm; *.slddrw)**.
- 3 Browse to the desired file, and click **Open**.

Filter Silhouette Edges


You can click **Filter Edges**  on the Selection Filter toolbar to filter silhouette edges. With the filter set, you can select only edges or silhouette edges.



Printing of Parts and Assemblies

You can print part and assembly documents to a specified scale. Earlier versions of the SolidWorks software only allowed printing to scale for drawing documents.

To print a part or assembly document at a specified scale:









- 1 Click **File, Page Setup**.
- 2 Under **Parts/Assemblies**, clear the **Print window** check box, set the **Scale** to the desired value, and click **OK**.
- 3 Click **Print**  or **File, Print**.

2D Command Emulator

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

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

Online Help

The online help interface is new in SolidWorks 99 *for users on Intel machines only*. Click **Help**  or **Help, SolidWorks 99 Help Topics** on the main toolbar to access the online help system. The help system features a browser that functions as a table of contents. Each  book icon represents a subject heading and each  page icon represents a help topic. Click   to expand the subject heading and click  to open and read a topic. Click  to find a keyword in the index, or click  to search all the words in the help file.

Reference Geometry

Projected Curve

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

To replace a sketch in a projected curve:

- 1 Right-click the **Curve** feature in the FeatureManager design tree, or right-click the curve in the graphics area, and select **Edit Definition**.
The **Project sketch onto sketch** dialog box appears.
- 2 Choose one of the following methods to clear the unwanted sketch from the **Sketch selected** box:
 - Click the sketch in the FeatureManager design tree.
 - Click the sketch in the graphics area.
 - Select the sketch in the **Sketch selected** box, and press **Delete**.
- 3 Select the replacement sketch in either the FeatureManager design tree or in the graphics area, then click **OK**.

Axis Normal to Surface at Point

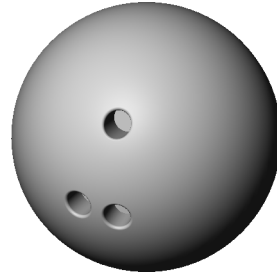
You can now create an axis normal to a *non-planar* surface at a point. In earlier versions of the SolidWorks software, you could create an axis normal only to a plane or a planar surface at a point.

Plane Tangent to Surface at Point


You can create a plane tangent to a non-planar surface at a point. The point does not have to be on the surface. The plane is created at a point either:

- On the surface closest to a selected sketch point.
- Projected onto the surface along the normal direction of the sketch plane of a selected sketch point.

For example, the sketches for the holes in the bowling ball are created on planes that are tangent to the surface of the bowling ball.



To create a plane tangent to a surface at a projected sketch point:

- 1 Open **Ball.sldprt**.
- 2 Click **Plane**  on the Reference Geometry toolbar, or **Insert, Reference Geometry, Plane**.
- 3 Click **On Surface** in the **Specify Construction Plane** dialog box, and click **Next**.

The **On Surface Plane** dialog box appears.

- 4 Click the surface of the sphere and one of the sketch points in the graphics area.
The preview of the plane appears, and it is tangent to the surface at the point closest to the selected sketch point.
- 5 Select the **Project sketch point along sketch plane normal** check box.
The preview of the plane appears, and it is tangent to the surface at the point where the sketch point is projected along the normal direction of the point's sketch plane.
- 6 Select the **Reverse direction** check box, if desired, to create the plane on the opposite side of the sketch plane.
- 7 Click **Finish**.

To complete the bowling ball:

- Repeat Steps 2 through 7 to create planes for the other two holes.
- Extrude circular cuts with a depth of 15mm from circles sketched on the three new planes. The circles are 10mm in diameter.
- Fillet the edges of the holes.

Import and Export

Import and Export Options

The import and export options are no longer located in the **Options** dialog box. They now appear on the **Open** dialog box for import, and on the **Save As** dialog box for export. When you select a file type that has import or export options, either the options appear on the dialog box, or an **Options** button appears. Click the **Options** button to access the options for that file type.

Changes made to the import and export options affect both the current document and the system defaults.

Other than the enhancements listed in the following sections, the options themselves have not changed.

IGES Files (.igs)

There have been a few enhancements to the IGES translator.

- The **Files of type** list in the **Open** dialog box now opens **.iges** files.
- NURBS** is now available as an export option for IGES output with trimmed surfaces.
- The **IGES Settings** dialog box no longer appears prior to IGES export. Because the export options are available from the **Save As** dialog box, the **IGES Settings** dialog box is no longer necessary.
- The software no longer attempts to knit blanked surfaces imported with an IGES file. Blanked surfaces are added to the FeatureManager design tree as surface features. In earlier versions of the SolidWorks software, an attempt was made by the software to knit blanked surfaces.

Import Diagnosis

You can diagnose and repair any gaps or bad faces on imported features. Import diagnosis works if only imported features are present in the FeatureManager design tree. Candidates for import diagnosis are:

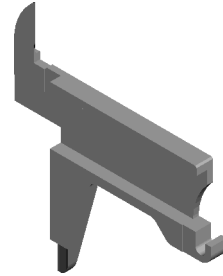
- An imported solid body that has rebuild errors.
- An imported surface that did not knit into a solid body.

To run an import diagnosis and to repair an imported feature:

- 1 Open **Import_diagnosis.sldprt**.

The part contains geometry imported from an IGES file that did not knit into a solid body because of problems in the imported geometry.

NOTE: This part was intentionally created with errors from a SolidWorks 98Plus sample file to illustrate **Import Diagnosis**. Normally, this sample file does not contain any errors.



- 2 Right-click the **Surface-Imported1** feature in the FeatureManager design tree, and select **Import Diagnosis**.

The **Import Diagnosis** dialog box appears.

- 3 Under **Gaps**, the message “**Import diagnosis: 2 gaps and 10 gap edges**” appears.
 - To view the gaps or gap edges, click **Gap** or **Gap edge**, then click **Previous** or **Next**.
 - To repair the gaps, click **Close All Gaps**.

The gaps are closed, and the message “**No gap found**” appears in the dialog box.

- 4 Click **OK**.

The geometry now forms a solid body, and **Surface-Imported1** is replaced by **Imported1** in the FeatureManager design tree.

DXF/DWG Mapping

DXF/DWG Mapping has undergone some changes for SolidWorks 99. The **SolidWorks To DXF/DWG mapping** dialog box changed to make layer creation, entity mapping, and color mapping simpler. In general, it is best to follow these rules when creating a map file:

- Color Mapping** takes precedence over entity mapping definitions.
- Take advantage of the **BYLAYER** property. If you do not assign a **Color** or **Line style** to an entity, the properties of the entity are the same as the properties of its layer.

To create a map file:

- 1 Open **Mapping.slddrw**.
- 2 Click **File, Save As**.
- 3 Select **Dxf Files (*.dxf)** in the **Save as type** list.
- 4 Click **Options**.
 - a) Select the **Custom Map SolidWorks to DXF** check box, if it is not already selected.
 - b) Clear the **Don't show mapping on each save** check box.
 - c) Click **OK** to close the **DXF/DWG Export Options** dialog box.
- 5 Type a name for the drawing file in the **File name** box, then click **Save**.
The **SolidWorks To DXF/DWG mapping** dialog box appears.

To add layers:

- 1 On the **Define Layers** tab, click **Add**.
The **Add New Layer Definition** dialog box appears.
- 2 Enter the information for the layer.
 - a) Type **GEOMETRY** (the name for the new layer) in the **Layer** box.
 - b) Select **Color 7** from the **DXF Colors** list.
 - c) Select **VISIBLE Normal** from the **Line Style** list.
 - d) Click **OK**.
- 3 Repeat Steps 1 and 2 to create three more layers with the following information:

Layer	DXF Colors	Line Style
NOTES	Color 7	<i>No selection*</i>
SECTIONLINES	Color 2	SECTION Thin
CROSSHATCH	Color 1	CROSSHATCH Thin

- 4 The four new layers are listed on the **Define Layers** tab.

* **Unspecified** appears under **Line style** for the **NOTES** layer. No line style mapping exists for entities that will appear on the **NOTES** layer.

To map entities:

- 1 Click the **Map Entities** tab, then click **Add**.
The **Add New Entity Mapping** dialog box appears.
- 2 Enter the entity mapping information:
 - a) Select **Geometry** from the **Entity** list.
 - b) Select **GEOMETRY** from the **Layer** list.
 - c) To take advantage of the **BYLAYER** property, do not select an item from the **DXF Color** list or the **Line Style** list.
 - d) Click **OK**.
- 3 Repeat Steps 1 and 2 to map three more entities with the following information:

Entity	Layer
Notes	NOTES
Section Lines	SECTIONLINES
Crosshatching	CROSSHATCH

The entity mappings are listed on the **Map Entities** tab.

- 4 Click **OK** to save the **.dxf** file.

DXF/DWG Import

If you import a **.dxf** or **.dwg** file with layers into the SolidWorks software, layers are created automatically in the drawing document. The layer information (names, properties, and entity locations) in the **.dxf** or **.dwg** file is retained.

To import a .dxf file with layers:

- 1 Open the **.dxf** file you created in the previous section. If you did not save the file, you can open **Mapping.dxf**.

The **Open DXF/DWG File** dialog box appears.

- 2 Click **OK** to accept the default values and open the file.

The **.dxf** file is imported to a SolidWorks drawing document. Four new layers are created in the drawing, one for each layer in the **.dxf** file. See **Line Formats and Layers** on page 5-8 for more information on controlling layers in a SolidWorks drawing document.

Sketching

Most SolidWorks features begin with a sketch. Becoming comfortable with using the sketch tools is, therefore, important. This chapter describes the enhancements to sketches, including:

- New sketch entities
- Pattern of sketch entities
- Offset of sketch entities
- Relations to sketch entities
- Edits to sketch entities
- 3D sketch


Creating Sketch Entities


Split Curve

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

You can dimension to a split point. Also, you can insert parts at split points in a piping assembly. For information about piping, see online help.

To split a sketch entity:

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

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

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

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


To combine two split sketch entities into one:

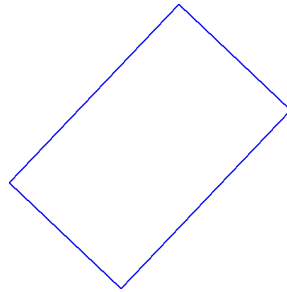
Click the split point, and press the **Delete** key.

Parallelogram


There is a new sketch tool, **Parallelogram**. This sketch tool allows you to create either a parallelogram or a rectangle whose sides are not horizontal or vertical with respect to the sketch plane.

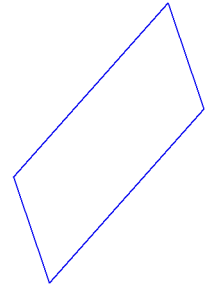
To create a rectangle at an angle:

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



To create a parallelogram:

- 1 In a part document, open a sketch.
- 2 Click **Parallelogram**  on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Parallelogram**.
- 3 Place the pointer where you want one corner of the parallelogram to appear.
- 4 Drag the pointer and release when one edge of the parallelogram is the correct length.
- 5 Hold down **Ctrl**, drag the pointer, and release when the parallelogram is the correct size and shape.





Duplicating Sketch Entities

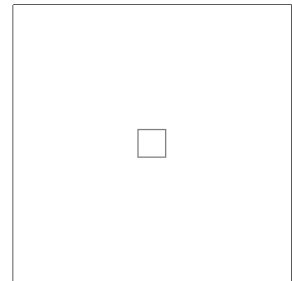
Arrays of Sketch Entities

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

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

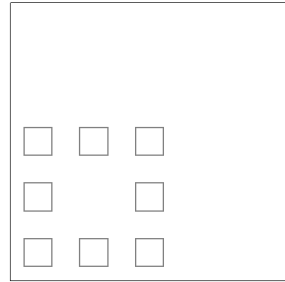
To create a linear array of sketch entities:

- 1 Open **Linear_sketch_array.sldprt**, and edit the **Square_hole** sketch.
- 2 Click **Linear Sketch Step and Repeat**  on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Linear Step and Repeat**.
The **Linear Sketch Step and Repeat** dialog box appears.
- 3 Click the **Items to repeat** box, then click the four sketch entities that make up the rectangle.
- 4 Under **Direction 1**:
 - Set **Number** to 3. This value includes the original entities.
 - Set **Spacing** to 20.00mm. This value is the distance from a point on one instance of the patterned sketch entities to the corresponding point on the next instance.
 - Leave **Angle** at 0°, then click **Reverse Direction 1**  to change the value by 180°.




NOTE: You can change the **Spacing** and **Angle** by dragging the selection point of the preview of the array.

- 5 To create a 2D array, repeat Step 4 for **Direction 2**.
- 6 Click **(2,2)** in the **Instances** box, then press **Delete** to delete this instance.
The instance **(2,2)** moves to the **Instances deleted** box.
- 7 Click **OK** to complete the array of sketch entities.




To create a circular array of sketch entities:

- 1 Open **Circular_sketch_array.sldprt**, and edit the **Circular_hole** sketch.
- 2 Click **Circular Sketch Step and Repeat**  on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Circular Step and Repeat**.

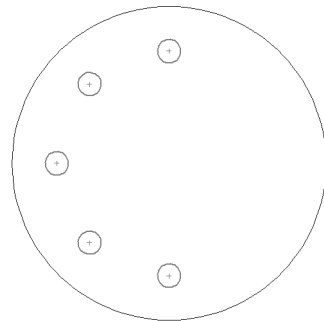
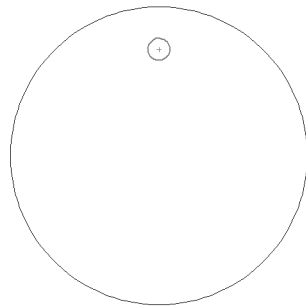
The **Circular Sketch Step and Repeat** dialog box appears.

- 3 Click the **Items to repeat** box, then click the circle.
- 4 To set the **Radius** and **Angle** values, drag the selection point in the center of the preview of the array and drop it on the sketch origin.
- 5 Under **Step**:

- Change **Number** to 5 and **Spacing** to 45°.
- Click **Reverse Rotation**  to change the pattern rotation to counterclockwise.

NOTE: You can change the **Radius**, **Angle**, and **Spacing** by dragging the selection point of the preview of the array.


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



Offset of Sketch Entities

You can offset sketch entities in the active sketch. In earlier versions of the SolidWorks software, you could only offset model edges or external sketch entities.

To offset sketch entities within the current sketch:

- 1 Click **Offset Entities**  on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Offset Entities**.
- 2 Select the entities that you want to offset.
- 3 In the **Offset Entities** dialog box:
 - Type a value in the **Offset** box.
 - Click the **Reverse** check box to change the offset direction.
 - Click the **Select chain** check box to select the entire contour.
- 4 Click **Apply**.
- 5 Click **Close** to offset the sketch entities and to exit the dialog box.

Selecting Sketch Entities

Automatic Inferencing Lines

You can turn off automatic inferencing lines while editing a sketch or while sketching in a drawing document.

If automatic inferencing lines are turned on, you no longer automatically infer to all model edges, active sketch entities, or non-active sketch entities. To infer to an entity, you need to activate the inferencing lines for the entity.

- In all documents, parts, assemblies, or drawings, you must activate the inferencing lines for model edges or non-active sketch entities.
- In parts or assemblies, the inferencing lines of the entities in the current sketch are always active.
- In drawings, you must activate the inferencing lines for current sketch entities.
- You must activate the inferencing lines for an entity each time you create a new sketch entity.

These changes improve sketching performance and reduce the chance of unwanted relations.

To turn off the automatic inferencing lines:

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

To activate the inferencing lines for an entity:


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

Silhouette Edges

Silhouette edges are no longer generated for an entire drawing view when the view is activated, nor for an entire model when the sketch is edited. Silhouette edges for a face are generated as the pointer hovers over the model face. This helps the performance of drawing view activation and sketch editing.

Editing or Moving Sketch Entities

Direct Edit
























You can specify the size and location of a sketch entity without dimensioning the sketch. If you have the **Enable direct edit of sketch** option turned on, the PropertyManager tab  appears when you select one or more sketch entities. (The PropertyManager tab replaces the **Properties** dialog box for sketch entities.)

NOTE: To fully define a sketch entity, you must still add relations or dimensions to the sketch entity. Any edits you make on the PropertyManager tab do *not* constrain a sketch entity.

To edit the properties of one or more sketch entities:

- 1 Click **Tools, Options, General**.
- 2 Select the **Enable direct edit of sketch** check box, and click **OK**.
- 3 Select one or more sketch entities in an active sketch.
The PropertyManager tab becomes active.
- 4 Modify the following properties of the sketch entities:
 - Select the **For construction** check box to change the selected entities to construction geometry.
 - If a spline is selected, select the **Proportional** check box to change the spline to a proportional spline.

5 Edit the following values, as necessary, for each sketch entity:

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

To move or copy one or more sketch entities:

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



No Solve Move

You can move sketch entities without solving the constraints (relations and dimensions) on the entities. If constraints exist between the selected entities and other sketch entities or model geometry, the following prompt appears after you drag the selected entities:

Model entities have constraints which involve other entities. Would you like to delete these constraints?

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

To turn on **No Solve Move**, use one of the following methods:

- Click **No Solve Move**  on the Sketch toolbar.
- Click **Tools, Sketch Tools, No Solve Move**.
- Select **No solve move** on the PropertyManager tab  as described in Step 3 of the previous section.

3D Sketch

You can create 3D sketches. A 3D sketch can be used as a sweep path, as a guide curve for a loft, or as one of the key entities in a piping system. For information about piping, see online help.

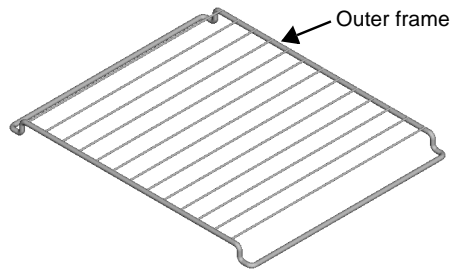
- ❑ You can add lines, points, centerlines, sketch fillets, and split curves to a 3D sketch.
- ❑ By default, you sketch relative to the default coordinate system in the model. To switch to one of the other two default planes, click the desired sketch tool, and press the **Tab** key. The origin of the current sketch plane is displayed.
- ❑ You can change the coordinate system for your 3D sketch. Click the desired sketch tool, hold down the **Ctrl** key, and click a plane, a planar face, or a user-defined coordinate system.
 - If you select a plane or a planar face, the 3D sketch planes rotate so that the XY sketch plane is aligned to the selected item.
 - If you select a coordinate system, the 3D sketch planes rotate so that the XY sketch plane is parallel to the XY plane of the coordinate system.
- ❑ You can dimension a 3D sketch.
 - You can add a length dimension by selecting two points, a line, two parallel lines, or a line and a plane or planar face.
 - You can add an angular dimension by selecting either three points or two lines.
- ❑ Relations are automatically added for sketch lines. As you sketch a line, the line snaps to one of the major directions, X, Y, or Z, if applicable, and is constrained as **Horizontal**, **Vertical**, or **Along Z**, respectively. The relations are added with respect to the current coordinate system for the 3D sketch.

You are not restricted to drawing lines along one of the three major directions. You can sketch in the current sketch plane at an angle to one of the major directions, or you can sketch out-of-plane if the endpoint of the line snaps to existing model geometry.

- ❑ You can add relations to points and lines in a 3D sketch. The following list shows some of the available relations.

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




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








To create the 3D sketch of the outer frame:

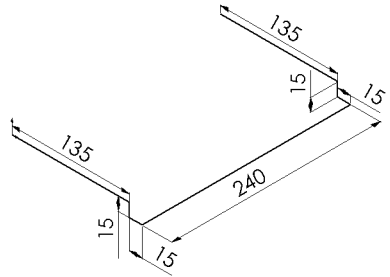
- 1 Open a new part, change the document's units to **Millimeters**, and change the view orientation to **Isometric**.

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

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

The pointer changes to  while sketching along the X axis.

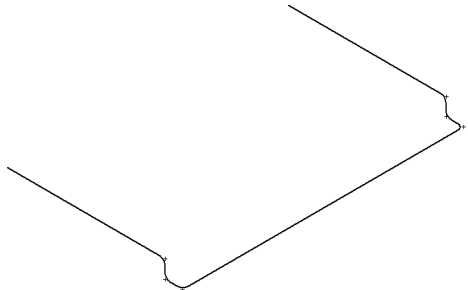
- 4 Continue sketching the other lines from the endpoint of the previous line. Each time you begin a new line, the origin for the current coordinate system is displayed at the beginning of the new line to help orient you as you sketch.
 - a) Sketch down the Y axis  for 15mm.
 - b) Sketch along the X axis for 15mm.
 - c) Press **Tab** to change the sketch plane to the YZ plane .
 - d) Sketch along the Z axis  for 240mm.
 - e) Press **Tab** twice to change the sketch plane back to the XY plane.
 - f) Sketch back along the X axis for 15mm.
 - g) Sketch up the Y axis for 15mm.
 - h) Sketch back along the X axis for 135mm.
- 5 Dimension each of the lines as shown.
- 6 Click **Fillet**  on the Sketch Tools toolbar, or click **Tools, Sketch Tools, Fillet** and fillet each intersection with a 5mm fillet.
- 7 Close the sketch.



The completed sketch is shown in the illustration.

To complete the outer frame:

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




Working with Parts

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


- Color edit
- Feature preview
- Feature creation
- Design tables
- Sheet metal
- Lighting

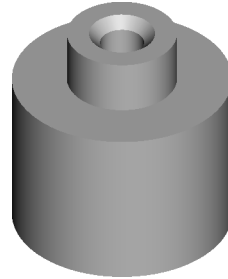
Editing Color or Sketches

Part Color and Appearance

You can change the color of selected entities faster in SolidWorks 99 than you could in earlier versions with the addition of the **Edit Color**  tool. You can change the color of an entire part, selected model faces, or selected features (including surfaces or curves).

To change the color of a part:

- 1 Open **Color.sldprt**.
Notice that the entire part is gray.
- 2 Click the part name at the top of the FeatureManager design tree, then click **Edit Color**  on the Standard toolbar.
- 3 Select a blue color from the color palette, then click **Apply**.
The entire part changes to the color blue.



To change the color of a face:

- 1 Click one of the outer circular faces of the base feature.
- 2 Select a green color from the color palette.
- 3 Make sure **Face** is selected under **Apply To**, then click **Apply**.
Only the selected face changes to the color green, not the entire feature, **Base**.


To change the color of a feature:

- 1 Click one of the faces of the hole in the center of the part.
- 2 Select a red color from the color palette.
- 3 Click **Feature** in the **Apply To** section, then click **Apply**.
The entire feature, **Center_cut**, changes to the color red.


To remove the color of a face:

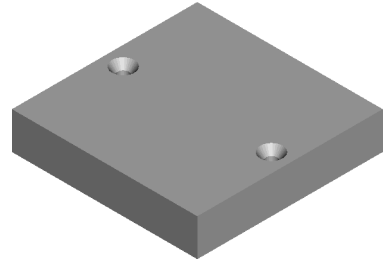
- 1 Click the green face on the model.
- 2 Make sure **Face** is selected under **Apply To**, then click **Remove**.
- 3 Click **OK**.
The face uses the body color, blue.

Dynamic Editing

You can see a preview of the resulting feature while editing a sketch that is used to create the feature. This preview is displayed for extrusions, revolves, sweeps, and lofts. To use this functionality, you must select the **Move/size features**  tool.

To dynamically update a feature while editing its sketch:

- 1 Open **Dynamic_update.sldprt**.
- 2 Click **Move/size features**  on the Features toolbar.
- 3 Click **Tools, Options, Performance**, and clear the **Dynamic update on sketch drag** check box.
- 4 Expand the **Screw_hole** feature in the FeatureManager design tree, then double-click **Hole_sketch** to edit the sketch.



Notice the preview of the feature is displayed while you edit the sketch.

- 5 Drag the centerline of the sketch toward the center of the model. When you drop the sketch entity, a new preview of the feature is displayed.
- 6 Click **Tools, Options, Performance**, and select the **Dynamic update on sketch drag** check box.
- 7 Drag the centerline of the sketch back to its original position.
Notice this time that the preview updates while you drag the sketch entity. This occurs because the **Dynamic update on sketch drag** check box is selected.
- 8 Drop the centerline when the hole is in the desired location, and exit the sketch.

TIP: You can also drag a sketch without editing the sketch. Double-click the feature in the FeatureManager design tree or the graphics area, then drag to move or size the feature.

Features

Pattern and Mirror


The linear pattern and circular pattern dialog boxes include:

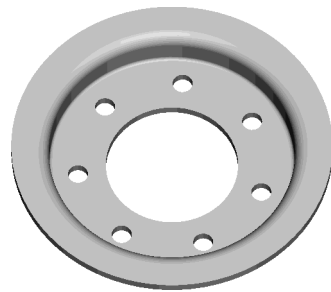
- A **Direction selected** box. The entity selected for the direction of the pattern is listed in this box.
- A **Preview** button. If you type in new values in the **Spacing** or **Total Instances** boxes, click the **Preview** button to update the preview.

Equal Spacing in Circular Patterns

You can create circular patterns of features where the pattern instances are equally spaced over a **Total Angle** value. The spacing between the pattern instances is determined from the total instances and the total angle for the circular pattern.

To create an equally spaced circular pattern:

- 1 Open **Flange.sldprt**.
- 2 Click the **Circular Pattern**  on the Features toolbar, or click **Insert, Pattern/Mirror, Circular Pattern**.
- 3 Click the **Direction selected** box, then select the temporary axis through the center of the flange.
- 4 Click the **Items to copy** box, then select the **Cut** feature.
- 5 Specify a **Total instances** value of **7**.
- 6 Select the **Equal spacing** check box.
The **Spacing** box changes to the **Total Angle** box.
- 7 Type **360°** in the **Total Angle** box
- 8 Click **OK** to create the pattern.



Edit Definition of Seed Feature

To edit the definition of the seed feature in a pattern, you can right-click a pattern instance in the graphics area. The pattern must be a pattern of only one feature.

Propagation of Seed Feature Color

The instances of a pattern or a mirror feature inherit the feature color of the seed feature as long as:

- the pattern or mirror is based on one feature.
- the color of the pattern or any faces on any pattern instances has not been changed.

Variable Radius Fillets

The method for creating a variable radius fillet has changed in SolidWorks 99. If you do not assign a value to a vertex, the radius is interpolated from the values assigned to the adjacent vertices. In earlier versions of the SolidWorks software, unassigned vertices used a constant default value.

Design Tables

There have been several improvements to design tables:

- You can create a new design table within a SolidWorks part or assembly document. Click **Insert, New Design Table** to insert a design table for the active document.
- You can delete configurations from the model that have been deleted from the design table. You are prompted when you close the table and given the choice to delete the configuration.
- You can add certain types of parameters to design tables by double-clicking the appropriate entity. With the appropriate worksheet cell active:
 - **Dimensions.** Double-click a dimension in the graphics area. (Make sure that the necessary dimensions are displayed *before* you insert the new design table.) The ***Dimension@Feature*** or ***Dimension@Sketch***<> parameter is inserted in the cell.
 - **Feature suppression.** Double-click a face of the feature. The ***\$STATE@Feature*** parameter is inserted in the cell.
 - **Component suppression.** Double-click a face of the component. The ***\$STATE@component<instance>*** parameter is inserted in the cell.
- To show design tables in drawing documents, select a drawing view of a model that contains a design table, and click **Insert, Design Table**. The design table appears on the drawing sheet just as it appears in the model.
- You can add header rows to the design table so you can display custom labels for the parameters. Hide the header rows that contain the names of the parameters, then add a row for the custom labels.
- You can edit a design table in a separate Excel window, instead of inside the SolidWorks window. Click **Tools, Options, General**, and select the **Edit design table in separate window** check box.

NOTE: To use design tables, we *strongly recommend* that you install Microsoft® Office Service Release 2 (SR-2).

To create a design table:

- 1 Open **Cap_screw.sldprt**.

The part document opens with its dimensions shown.

- 2 Click **Insert, New Design Table**.

A new design table appears for the part document.

- 3 In row **2**, enter the dimensions that you want to control:

- a) Click cell **B2**, if it is not already selected.

- b) Double-click the **0.7500** diameter dimension in the graphics area.

The dimension name appears in cell **B2**, the dimension value appears in cell **B3**, and cell **C2** is now the active cell.

- c) Repeat the previous step for the other dimensions in the following order: **1.2500**, **0.5460**, **0.1490**, and **0.3380**.

- 4 In column **A**, cells **A3** through **A6**, enter the configuration names **001-7500**, **001-6250**, **001-5625**, and **001-5000** (overwrite **First Instance** in cell **A3**.)

- 5 Hide the header rows, then enter labels for the design table dimensions:

- a) Select rows **1** and **2**, then right-click and select **Hide**.

- b) Select row **3**, and click **Insert, Rows**.

- c) In cells **B3** through **F3**, enter the labels: **E**, **A**, **H**, **J**, **T**.

- 6 Enter the dimension values as shown:

	E	A	H	J	T
001-7500	0.7500	1.2500	0.5460	0.1490	0.3380
001-6250	0.6250	1.0000	0.4370	0.1330	0.2700
001-5625	0.5625	0.9370	0.4090	0.1180	0.2520
001-5000	0.5000	0.8120	0.3540	0.1060	0.2180

- 7 Adjust the column width, row height, text alignment, borders, fonts, and so on, as needed to meet your company's standards. The appearance of the design table in the model will be the same when you insert the design table in a drawing document.

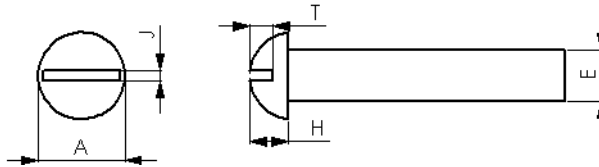
- 8 Click outside the table in the graphics area to close the design table.

To display a design table in a drawing:

- 1 Open **Cap_screw.slddrw** while the **Cap_screw.sldprt** document is still open.
- 2 Select either drawing view, and click **Insert, Design Table**.
The design table appears, and you can drag it into place on the sheet.
- 3 If you need to change the size of the design table, you can modify the properties of the design table in the same manner as other OLE objects. For more information, see **Scaling of OLE Objects** on page 1-3.
- 4 Modify the dimension names to match the labels in the design table:
 - a) Right-click a dimension in the drawing, and select **Properties**.
 - b) Click **Modify Text**.
 - c) Delete the existing text from the middle line of the **Modify Text of Dimension** dialog box, then enter the corresponding label from the table.
 - d) Repeat for each of the dimensions.

The completed drawing is shown in the following illustration:

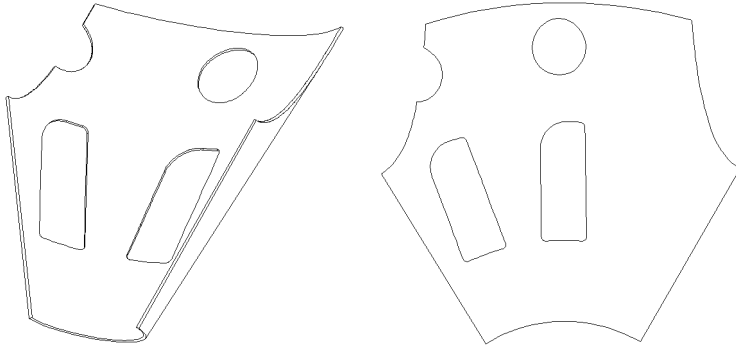
	E	A	H	J	T
001-7500	0.7500	1.2500	0.5460	0.1490	0.3380
001-6250	0.6250	1.0000	0.4370	0.1330	0.2700
001-5625	0.5625	0.9370	0.4090	0.1180	0.2520
001-5000	0.5000	0.8120	0.3540	0.1060	0.2180



Sheet Metal

Conical and Cylindrical Faces


You can unroll sheet metal parts containing conical faces. The following illustration shows a conical part in the bent and flattened states.



Several limitations that previously existed for sheet metal parts containing cylindrical faces have been eliminated.

- Cylindrical bends can have more than four bounding edges. However, at least one end of the cylindrical face must have at least one linear edge.
- Cylindrical bends can have beveled edges.

Fixed Edge or Face

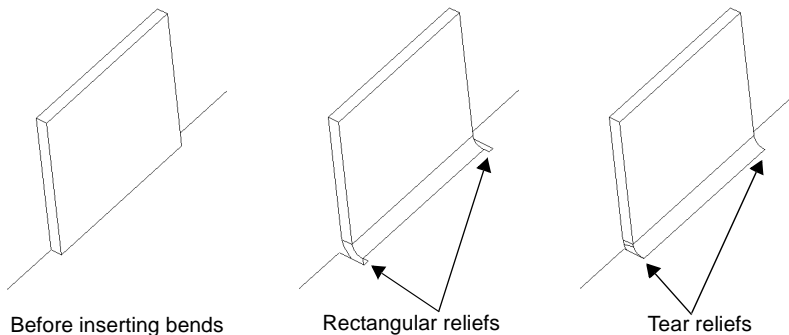
You no longer have to pre-select a face or edge prior to clicking **Insert Bends** . You can also change the fixed entity when editing the definition of a **Sheet-Metal** feature.

Relief Ratio

The **Offset ratio** box in the **Auto Relief** section of the **Flatten-Bends** dialog box has been renamed to **Relief ratio**. The functionality is the same; only the label has changed.

Relief Cuts

Tear reliefs are a new type of relief cut, in addition to rectangular reliefs. With a tear relief, the model tears at the bend region. The following illustration compares tear reliefs and rectangular reliefs.



You can modify the type of relief cuts automatically added to an entire sheet metal part or to each bend.

To change the auto relief type for all bends:

- 1 Right-click the **Sheet-Metal** feature in the FeatureManager design tree, and select **Edit Definition**.
- 2 In the **Auto Relief** section, change the type of relief cut.
- 3 If you change the type of relief cut to **Rectangular**, enter the **Relief ratio** value.
- 4 Click **OK**.

To change the auto relief type or size for a individual bend:

- 1 In the FeatureManager design tree, click the beside the **Flatten-Bends** feature.
- 2 Right-click the bend, and select **Edit Definition**.
- 3 Click the **Auto Relief** tab, then click to clear the **Use default relief** check box.
- 4 Change the type of relief cut in the **Relief Type** section.
- 5 If you change the type of relief cut to **Rectangular**, enter values for the **Width** and the **Depth**.
- 6 Click **OK**.

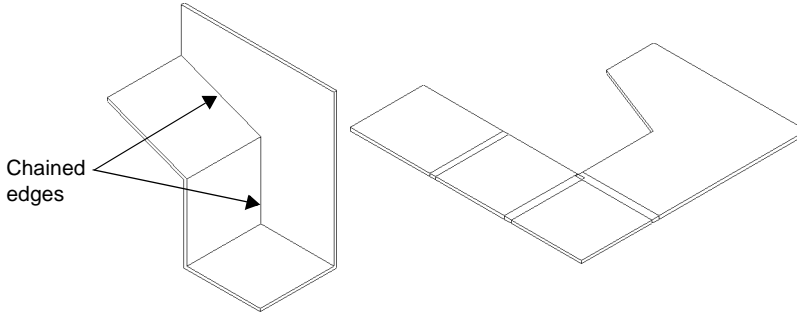
NOTE: You can only edit the relief type of an individual bend on sheet metal parts created in SolidWorks 99.

Rip Feature


You can rip the inside linear edges of adjacent planar faces in a sheet metal part so that you can flatten the part.

- The adjacent faces must form one or more linear edges or a chain of linear edges.
- Chained edges must be adjacent to a common planar face.

The following illustration shows a model to which you can add a **Rip** feature. The model is shown before inserting bends and in the flattened state.

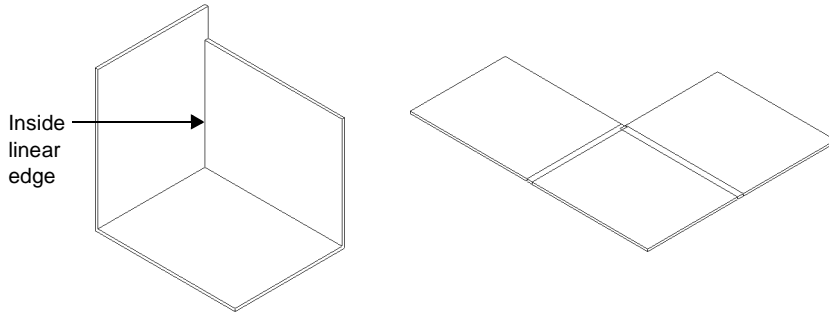



To create a sheet metal part using rips:

- 1 Open **Rip_feature.sldprt**.
- 2 Click **Insert Bends**  or **Insert, Features, Bends**.
The **Flatten-Bends** dialog box appears.
- 3 On the model, select the fixed face.
The name of the fixed face appears in the **Fixed edge or face** box.
- 4 Specify a **Default bend radius** value of **2mm**.
- 5 Select **Tear** in the **Auto Relief** list. (You can use **Rectangular** relief cuts with the **Rip** feature, but this example uses a **Tear** relief cut.)

- 6 Click the **Rip** tab.
- 7 On the model, select the *inside linear edge* to rip.
- 8 Click **OK**.



The following illustration shows the part before inserting bends and in the flattened state:




NOTE: You can create rips in any part of uniform thickness before inserting bends to create a sheet metal part. Click **Insert Rip**  on the Features toolbar, or click **Insert, Features, Sheet Metal, Rip**. Then insert the bends to create the sheet metal part.

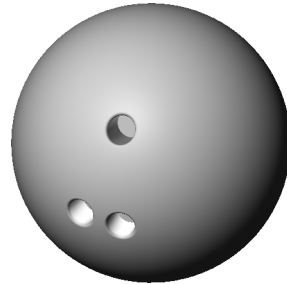
Lighting

There are several enhancements to lighting in SolidWorks 99, such as:


- ❑ There is a **Lighting** folder  in the FeatureManager design tree. The **Lighting** folder  contains each of the light sources in the model.
- ❑ There is a new light type, **Point** light. A **Point** light comes from a very small light source located at a specific coordinate in the model space. This type of light source emits light in all directions. The effect is like a tiny light bulb floating in space.
- ❑ As you edit each light, a graphical representation of the light source is displayed, indicating the color and direction of the light.
- ❑ Each light type has its own dialog box with its own properties. In earlier versions, all of the light types shared the same dialog box.

To add a point light:

- 1 Open **Ball_complete.sldprt**.
- 2 Right-click the **Lighting** folder  in the FeatureManager design tree and select **Add Point Light**, or click **Insert, Light, Point Light**.

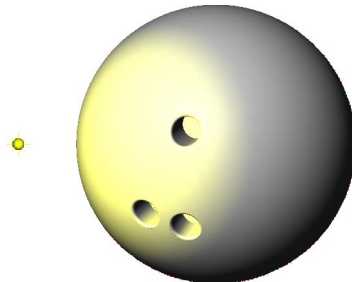


To edit the properties of a point light:

- 1 Expand the **Lighting** folder  in the FeatureManager design tree.
- 2 Right-click **Point1**, and select **Properties**.
The **Point Light Properties** dialog box appears.
- 3 To change the color of the light, click **Edit**, select a yellow color from the palette, and click **OK**.
- 4 To turn the light on or off, select or clear the **On** check box.
- 5 If you want to change the intensity of the light, edit the **Ambient**, **Brightness**, or **Specularity**.
- 6 Click the **Position** tab.
- 7 Specify a **Distance** of **100mm** and a **Latitude** of **135°**.

The point light appears outside the bowling ball, casting a yellow glow onto the bowling ball.

- 8 Click **OK**.




Working with Assemblies

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


- Assembly color
- Curvature display
- Reload of a component
- Assembly structure editing
- SmartMates and mate references
- Collision Detection

General Information

Assembly Color and Appearance

By default, components that you add to an assembly are displayed using the color specified in the original part document. You can override the part color for selected instances, or use the default color for the assembly. You can change the color more easily in SolidWorks 99 than you could in earlier versions with the addition of the **Edit Color**  tool.

To change the color of selected component instances:

- 1 Open `\Reorganize_assy\Ujoint.sldasm`.
- 2 Hold down **Ctrl** and click **U-joint_pin1<1>**, **U-joint_pin2<1>**, and **U-joint_pin2<2>** in either the FeatureManager design tree or the graphics area.
- 3 Click **Edit Color**  on the Standard toolbar, select a red color from the color palette, then click **OK**.

All three components change to the color red.

Curvature Display

You can now display an assembly document with the surfaces rendered in different colors according to the local radius of curvature.

NOTE: This process can take a long time for models with complex surfaces.

To render curved surfaces in colors:

Click **View, Display, Curvature**.

To render an individual model face in colors:

Right-click the model face, and select **Face Curvature**.

Reload/Replace

You can now replace a document with a document of the same name but in a different folder. **Replace** lets you replace each open reference to the current document with a reference to a new document. For more information on **Replace**, refer to the online help.

Assembly Structure Editing

There are several ways to edit the structure of an assembly document:

- ❑ You can reorder the components within the FeatureManager design tree. The new order is reflected in any bill of materials of the assembly.
- ❑ You can insert a new, empty sub-assembly at any level of the assembly hierarchy while you are editing a top-level assembly, then add components to it in a variety of ways.
- ❑ You can form a new sub-assembly by selecting a group of components that are already in the assembly. This creates a sub-assembly and adds components to it in a single step.
- ❑ You can *dissolve* a sub-assembly into individual components, thereby moving the components *up* one level in the assembly hierarchy.
- ❑ You can move selected components up or down within their own branch of the assembly hierarchy, or move them laterally, to a sub-assembly in a different branch in the assembly hierarchy.

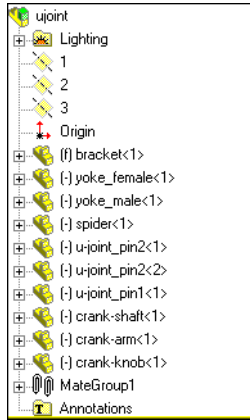
When you dissolve a sub-assembly, or reorganize the components at any level, the mates and any features that reference the selected components are affected. Therefore, decisions about hierarchical groupings should be made early in the development of a complex assembly to minimize the effects on these items. Some of these effects include:

- ❑ You cannot move a component with in-context features. Any component that has external references to another component in the assembly cannot be moved into a different assembly.
- ❑ Parts created in context, including weld beads or envelope components, cannot be moved into a different assembly.
- ❑ Mates move to the mategroup of the lowest common parent assembly.
- ❑ Explode steps and assembly feature cuts and holes are deleted.
- ❑ Equations may not solve because the instance number suffix $\langle n \rangle$ often changes when you reorganize the components. This is done to prevent conflicts with existing components in the destination assembly.

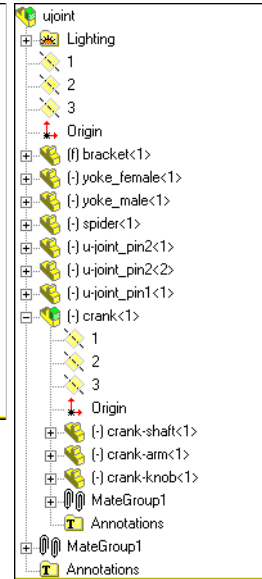
To form a new sub-assembly from existing components:

- 1 Open `\Reorganize_assy\Ujoint.sldasm`.
Notice that **Crank-shaft<1>**, **Crank-arm<1>**, and **Crank-knob<1>** are components of the **Ujoint** assembly. Since these individual components make up the crank that turns the assembly, it is a good idea to combine these components into a new sub-assembly.
- 2 In the FeatureManager design tree, hold down the **Ctrl** key and select **Crank-shaft<1>** and **Crank-arm<1>**.
- 3 Right-click one of the selected components, and select **Form New Subassembly Here**.
The **Save As** dialog box appears.
- 4 Enter **Crank.sldasm** in the **File name** box, and click **Save**.

Without Sub-assembly




With Sub-assembly



The new sub-assembly, **Crank**, is inserted at the level where the components were located, and the selected components are added to the new sub-assembly. Mates that exist between the selected components move from the mategroup of **Ujoint** to the mategroup of **Crank**.


To move a component to a sub-assembly:

Click **Crank-knob<1>** in the FeatureManager design tree, then drag it on top of the sub-assembly, **crank**. Look for the  pointer.

Crank-knob<1> is now a component of the sub-assembly, **Crank**.



NOTE: You can also reorganize assembly components by clicking **Tools, Reorganize Components**. For more information about reorganizing components in this manner, see online help.

To reorder a component:

Click **Spider<1>** in the FeatureManager design tree, then drag it on top of the **Bracket<1>** component. Look for the  pointer.

Spider<1> now appears in the FeatureManager design tree below **Bracket<1>**.

To reorder a component below a sub-assembly component:

- 1 Click **Spider<1>** in the FeatureManager design tree, then drag it to the **Crank<1>** sub-assembly component. The pointer changes to .
- 2 Hold down **Alt** to change the pointer to . Drop **Spider<1>**.
Spider<1> now appears in the FeatureManager design tree below **Crank<1>**.

To dissolve a sub-assembly:

Right-click **Crank<1>** in the FeatureManager design tree, and select **Dissolve Sub-assembly**.

The components of the sub-assembly, **Crank**, become components of **Ujoint**, and **Crank** is removed. Mates that used to exist in the mategroup of **Crank** move to the mategroup of **Ujoint**.



SmartMates and Mate References

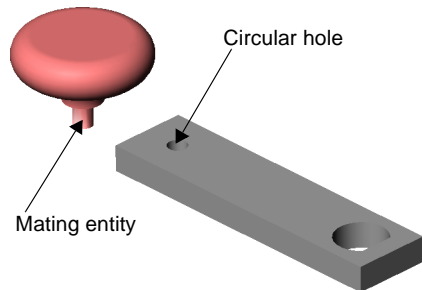
Inferring Mates While Moving Components

You can add SmartMates when you move a component in an assembly, even if the component is used in other mates. As you drag the component into place, you can infer potential mate partners for creating geometry-based SmartMates.

NOTE: If a component is already used in other mates, you can only move it within the degrees of freedom allowed by those mates.

To create SmartMates while moving a part:

- 1 Open **Smart_mates.sldasm**.
- 2 Click **Smart Mates**  on the Assembly toolbar.
- 3 Double-click the mating entity as shown.
- 4 Drag **crank-knob** onto **crank-arm** until the pointer is over the circular hole on **crank-arm**.
The pointer changes to .
- 5 Drop **crank-knob** to add a **Concentric** mate between the two faces.



When you move a component with a mating entity selected, the component is displayed with a transparent appearance. This indicates that you can infer potential partners for SmartMates while dragging.

To display the component with its normal (not transparent) appearance, hold the **Shift** key as you drag. This can help you see what the assembly will look like when the mate is completed. To return to the transparent appearance, release the **Shift** key.

To temporarily suspend the inferring of SmartMates while dragging, press the **Alt** key. You can then drag the component in the normal way. When you want to infer potential mate partners again, press **Alt** again.

Mate References




You can now take advantage of SmartMates when you add parts by dragging them from the Feature Palette window or from Windows Explorer. Add a mate reference to identify which entity of a part is to be used for SmartMates. When you insert a part with a mate reference by dragging its part icon from the Feature Palette window or from Windows Explorer, the software can identify potential mate partners for the specified entity.

Collision Detection

You can detect collisions with other components when moving or rotating a component. You can detect collisions with the entire assembly or a selected group of components. You can find collisions for just the moving component or for all components that move as a result of mates to the selected component.

The SolidWorks software can only detect collisions on the following surface types: *planar*, *cylindrical*, *conical*, *spherical*, and *torii*. If there are other surface types in the scope of the Collision Detection, the number of undetectable surfaces appears in the PropertyManager tab.

To detect collisions:

- 1 Open **Smart_mates.sldasm**.
- 2 Create another copy of the **crank-knob** component in the assembly. (One way to do this is to hold down **Ctrl**, then drag the **crank-knob** from the FeatureManager design tree to the graphics area.)
- 3 Click **Move Component**  or click **Tools, Component, Move**.
The PropertyManager tab becomes active.
- 4 Click the **Collision Detection**  check box.
- 5 To stop the component's movement when a collision is detected, click the **Stop at collision** check box.
- 6 Make sure that **All components** is selected under **Scope of check**.
With **All components** selected, collisions with all assembly components are detected. If you choose **Selected**, only collisions with a group of assembly components that you select are detected.
- 7 Make sure **Component being dragged** is selected under **Find collisions for**.
With **Component being dragged** selected, only collisions on the moving component are detected. If you choose **Entire scope**, collisions are detected for both the moving component and any components that move as a result of mates to the moving component.
- 8 Click the **crank-knob** in the graphics area and drag it around.
The **crank-knob** stops any time your drag causes the component to collide with one of the other components in the assembly. The faces that collide are highlighted.
- 9 Click **Move Component**  again to stop moving the component.

Drawings

You can create 2D drawings of the 3D solid parts and assemblies you design. This chapter describes the enhancements to drawings, including:

- Printing drawings
- Performance improvements
- Drawing views
- Dimensions
- Annotations
- Bill of materials
- Line formats and layers

General Information

Print Drawings in Color

You can print and print preview a drawing in color if the selected printer supports color. If the printer does not support color, the print and the print preview are both in black and white.

To print a drawing in color:

- 1 Select a printer that supports color.
- 2 Click **File, Page Setup**.
- 3 Select the **Print drawings in color** check box to turn on color printing, and click **OK**.

Performance Improvements

The performance of detail views and section views is better in SolidWorks 99, especially for views of assembly documents.

Silhouette edges are no longer generated for an entire drawing view when the view is activated, nor for an entire sketch when the sketch is edited. Silhouette edges for a face are generated as the pointer hovers over the model face. This helps the performance of drawing view activation and sketch editing.

Drawing Rulers

You can turn off the rulers that appear in the graphics area of a drawing.

To turn off the drawing rulers:

Click **View, Rulers** to turn off the display of the rulers.

Design Tables

Design tables can be shown in drawings. The design table appears on the drawing sheet just as it appears in the model. For more information, see **Design Tables** on page 3-5.

Drawing Views

Adding Drawing Views

There are new methods for adding drawing views. You can:

- Add drawing views of an assembly component from an open assembly document.
- Add drawing views of a part or assembly by clicking an existing view in a drawing document.
- Add drawing views of a part or assembly from an unopened file.

To add a drawing view of an assembly component from an open assembly document:

- 1 In a drawing document, click **Insert, Drawing View**, then click **Relative To Model, Named View**, or **Standard 3 View**, as desired. You can also click the corresponding buttons on the Drawing toolbar.

The pointer changes to indicate that you need to select a model.

- 2 Switch to an open assembly document window.
- 3 Select a face on a part in the graphics area, or click the name of either the individual part or the sub-assembly in the FeatureManager design tree.

The placement of the views on the drawing sheet behaves the same as if you had selected a part or assembly document.

To add a drawing view of a part or assembly from an existing drawing view:

- 1 In a drawing document, click **Insert, Drawing View**, then click **Relative To Model, Named View**, or **Standard 3 View**, as desired. You can also click the corresponding buttons on the Drawing toolbar.

The pointer changes to indicate that you need to select a model.

- 2 Click a drawing view in the same drawing or in a different drawing document that contains the desired part or assembly, either in the FeatureManager design tree or in the graphics area.

The placement of the views on the drawing sheet behaves the same as if you had selected a part or assembly document.

To add a drawing view from a unopened file:

- 1 In a drawing document, click **Insert, Drawing View**, then click **Relative To Model, Named View**, or **Standard 3 View**, as desired. You can also click the corresponding buttons on the Drawing toolbar.

- 2 Right-click and select **Insert From File**.

The **Insert Component** dialog box appears.


- 3 Browse to the desired part or assembly document, and click **Open**.

The placement of the views on the drawing sheet behaves the same as if you had selected a part or assembly document.

Rotating Drawing Views

You can rotate a view around its center point to orient the view at any angle.

To rotate a drawing view around its center point:

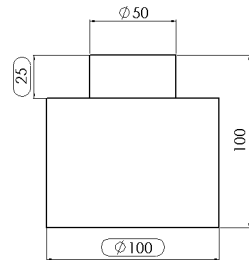
Click **Rotate View**  on the View toolbar, then drag the view to the desired rotation. You can also right-click a view, and select **View, Rotate View**. The view snaps to 45° increments, but you can drop the view at any angle.

To restore the view to its original rotation, right-click the view and select **Alignment, Default Alignment**. If the default alignment to another view was broken, this also restores the original alignment.

Dimensions

Inspection Dimensions

You can convert any existing dimension into an inspection dimension. Right-click a dimension, and select **Show as Inspection** to display a dimension as an inspection dimension.



Ordinate Dimensions


The **Single command per pick** check box on the **Options** dialog box is now ignored when creating ordinate dimensions.


Newly placed ordinate dimensions automatically jog to avoid overlapping text. The size of the jog is determined by the size of the text and the potentially overlapping area of the text.

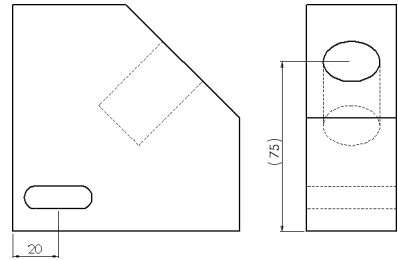
New Selections for Dimensions

You can dimension to the midpoint of a linear model edge in a drawing view. Also, you can dimension to the center of a circle that is projected as an ellipse in the drawing view.

To dimension to the midpoint of an edge or to a circle projected as an ellipse:

- 1 Open **New_dimensions.slddrw**.
- 2 Activate the front drawing view.
- 3 Click **Dimension** , or right-click and select **Dimension**.
 - a) Select the left edge of the model.
 - b) Right-click a linear edge of the slot, and click **Select Midpoint**.
 - c) Click to place the dimension.


- 4 Press **Esc** to turn off the dimension tool.
- 5 Activate the right drawing view.
- 6 Click **Dimension** , or right-click and select **Dimension**.
 - a) Select the bottom edge of the model.
 - b) Select the edge of the circular hole (projected as an ellipse in this view).
 - c) Click to place the dimension.

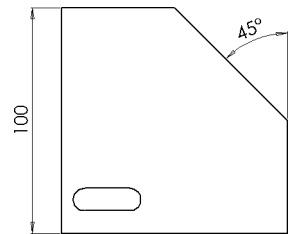


Display of Dimensions


You can override a drawing document's standard to change the display of dimensions. You can do this for linear, angular, or diameter dimensions. You can change all of the dimensions of one type in a drawing, or individual dimensions.

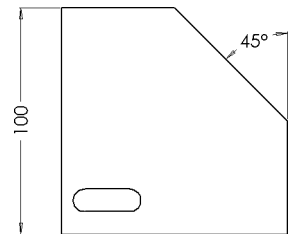
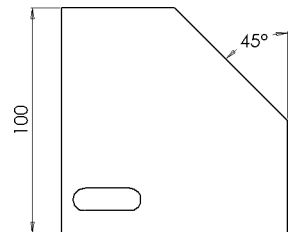
To change the display of all of the dimensions of one type in a drawing:

- 1 Open **Witness_display.slddrw**.
- 2 Click **Tools, Options, Detailing**, then click **Leaders**.
The **Dimension Leaders** dialog box appears.
- 3 Select the **Override Standard's Leader Display** check box, then click  to change the appearance of angular dimensions to a broken leader with horizontal text.
- 4 Click **OK** twice.
The display of the angular dimension changes.



To change the display of a dimension on a drawing:

- 1 Right-click the **100mm** dimension, and select **Properties**.
The **Dimension Properties** dialog box appears.
- 2 Click **Display**.
- 3 Click  to change the appearance of this linear dimension to a broken leader with aligned text.
- 4 Click **OK** twice.
The display of the linear dimension changes.



Annotations

Blocks

Custom symbols are now called blocks. You can scale or rotate blocks in the drawing.

To scale or rotate a block in a drawing:

- 1 In a drawing, click the sheet, view, or model where you want to add the block, then click **Insert, Annotations, Block**.
- 2 Browse to the desired block document, and click **Open**.
The block appears on the drawing.
- 3 Double-click the block, or right-click the block and select **Properties**.
- 4 To scale the block, enter a value in the **Scaling** box, or move the **Scaling** slider. Move the slider to the right to increase the scale value.
- 5 To rotate the block, enter a value in the **Rotation** box, or move the **Rotation** slider. Move the slider to the right to rotate the block counterclockwise.
- 6 Click **OK** to close the **Properties** dialog box.

Notes

You can now have more than 1024 characters in a note.

Bill of Materials

There are several enhancements to bill of materials in SolidWorks 99. You can:

- Select which template to use as you create a bill of materials.
- Edit the properties of a bill of materials.
- Control the visibility of items (or rows) in the bill of materials.
- Control the appearance of a row in the bill of materials if a component is deleted from the assembly.
- Override the item numbers assigned by the software, and then assign new item numbers in Excel.
- Move the labels from the top of the bill of materials to the bottom.
- Modify the item numbers in the bill of materials to override the default order.
- Split the bill of materials if the table exceeds a height that you specify.

NOTE: To use bill of materials, we *strongly recommend* that you install Microsoft Office Service Release 2 (SR-2).

To insert a bill of materials:

- 1 Open **Bill_of_materials.slddrw**.
- 2 Select the drawing view, and click **Insert, Bill of Materials**.
The **Select BOM Template** dialog box appears.
- 3 Select the default template **Bomtemp.xls**, and click **Open**.
The **Bill of Materials Properties** dialog box appears.
- 4 Click **OK** to create a bill of materials with the default selections.
The bill of materials appears as a single-width table with nine items.
- 5 Drag the bill of materials to the desired location on the drawing sheet.

To edit the contents of a bill of materials:

- 1 Right-click the bill of materials, and select **Properties**, or select the bill of materials, and click **Edit, Properties**.
The **Bill of Materials Properties** dialog box appears.
- 2 On the **Contents** tab, click to clear the check mark next to **crank-shaft**, **crank-arm**, and **crank-knob** to exclude those items from the bill of materials.
- 3 Clear the **Display labels at top** check box to display the labels at the bottom of the bill of materials, if desired.
- 4 Click **OK**.
The bill of materials appears as a single-width table with only six items.

To split the bill of materials:

- 1 Edit the properties of the bill of materials using one of the methods from Step 1 of the previous section.
- 2 On the **Control** tab, select the **Split tables into multiple sections** check box.
- 3 Specify a **Table height before splitting** value of **25mm**. Once the height of the bill of materials exceeds this value, the table splits.
- 4 Make sure **Right** is selected in the **Split direction for new sections** box.
- 5 Click **OK**.

The bill of materials appears as a double-wide table with a total of six items.

Line Formats and Layers

Line Formats

You can specify the color, thickness, and style that is used for any sketch entities that you add to a drawing. You can also modify the appearance of existing lines, edges, and sketch entities in a drawing.


Because you can place entities on layers, **By layer** is the default selection for each tool on the Line Format toolbar. If **By layer** is selected, the properties of the entity are the same as the properties of the active layer. If **By layer** is cleared, the properties of any entities you create will be different from those specified for the layer.

Layers

You can create layers in a SolidWorks drawing document. You assign a line color, line thickness, and line style for new entities created on each layer. New entities are automatically added to the active layer. You can hide or show individual layers.

Dimensions, annotations and sketch entities can be moved onto layers. Dimensions and annotations use the color specified for the layer. Sketch entities use all the properties of the layer.

To create drawing layers and move existing entities to layers:

- 1 Open **Cap_screw.slddrw**.
- 2 Click **Layer properties**  on the Drawing toolbar.
The **Layers** dialog box appears.
- 3 Click **New**, and enter **DIMENSIONS** as the **Name** for the new layer.
DIMENSIONS becomes the active layer.


- 4 Click the **Color** box, select a red color from the palette, and click **OK**.
- 5 Hold down **Ctrl**, select the dimensions in the drawing, then click the **Move** box for the **DIMENSIONS** layer.

The dimensions move to the selected layer, **DIMENSIONS**, and change color to red.



To hide and show a layer:

- 1 Click the yellow light bulb for the **DIMENSIONS** layer.
The light bulb turns gray, and all the entities on the layer are hidden.
- 2 To show the layer, click the light bulb again.
- 3 Click **OK**.

To sketch entities and override the layer's properties:

- 1 Click **Line** , sketch a line on the drawing sheet, then press **Esc** to turn off the line tool so you can see the color of the new line.

The line appears in a red color, the color of the layer.

- 2 Click **Line Color**  on the Line Format toolbar.
- 3 Select a blue color from the palette, and click **OK**.
- 4 Click **Line** , sketch another line on the drawing sheet, then press **Esc**.

The line appears in a blue color instead of the layer color.

NOTE: Both lines are on the active layer, **DIMENSIONS**, even though they have different colors.

You can create a special type of sub-assembly that builds a routing path between components. A routing sub-assembly is made from components with certain required features and properties. To specify the geometry of the path, you use a 3D sketch.

This chapter describes:

- Connection points and Route points
- Creating the necessary parts
- Routing options
- Creating and editing a routing sub-assembly

The piping functionality is supplied as an add-in application. To enable the functionality, click **Tools, Add-Ins**, and select the **Piping** check box.

Overview

A routing sub-assembly is always a top-level assembly component. Whenever you insert a component with routing information into an assembly, a routing sub-assembly is created for you. Unlike other types of sub-assemblies, you do *not* create a routing assembly in its own window, and then insert it as a component in the higher-level assembly.

The SolidWorks software supports routing functionality for fabricated pipes. You model the path of the pipe by creating a 3D sketch of the pipe centerline. The software uses the centerline definition to generate the pipe and elbow components for the route.

The software makes extensive use of design tables to create and modify the configurations of routing components. The configurations are distinguished by different dimensions and properties. If you are unfamiliar with these concepts, see Design Tables in the online help.

A pipe part contains a configuration for each type and size of raw stock. As you create and edit the route, a new configuration is generated automatically for each unique cut length of the selected stock. The configurations are saved in a new pipe part; the original pipe part in the library folder is not changed.

Where there are bends in the path, elbows are added automatically. You specify a default elbow fitting to be used at each bend in the route. If you want, you can also let the software create custom elbows at bends whose angle is different from the standard bend angle.

You can add various types of fittings to the route, such as flanges, tees, crosses, and reducers. The fitting components must have configurations that match the pipe sizes.

General Procedure

The following describes a general procedure for developing the route for a pipe. A more detailed description can be found on page 6-14.

- 1 Create the part documents for the pipe and the fittings (elbows, flanges, and so on).
- 2 Create an assembly with the components to be connected (tanks, pumps, and so on). Add any other components that are needed to specify the path of the route. This includes attachment components such as supporting brackets, obstacles that the routing path must avoid, and so on.
- 3 Position the fixed components using mates, dimensions, or relations to a layout sketch, and save the assembly.
- 4 Insert the necessary end fittings, and mate them to the components you need to connect. A new routing sub-assembly containing the end fittings is created.
- 5 Name the new routing sub-assembly, and specify the route properties (pipe part, elbow part, and other options as described later in this chapter).
- 6 Create a 3D sketch to specify the path. The 3D sketch should include sketch fillets at sharp corners (virtual sharps of two straight segments) and any other bend locations that require elbows.

7 Add tees, crosses, and so on, as needed.

When you exit the 3D sketch, the software generates the pipe segments, inserts the elbows, and connects all the components into a continuous pipe assembly.

Each component in the routing assembly is parametrically related to the 3D sketch. If you change the sketch in any way, the pipes and fittings are updated automatically.

You do not need mates or other relations between the components of the routing sub-assembly, because the sizes and positions are driven by the 3D sketch.

3D Sketch

You use a 3D sketch to define the centerline of the route path.

When you drag a flange into an assembly, a new routing sub-assembly is created, and the 3D sketch is started automatically. You can also begin the 3D sketch manually, if the fittings are already in the assembly. Right-click the connection point in a fitting and select **Start Route**.

Sketch the lines, and add fillets as needed. If you select the option **Automatically create sketch fillets**, a fillet is added at each virtual sharp intersection, using the default radius specified in the **Route Properties**.

For information about creating a 3D sketch, see **3D Sketch** on page 2-9.

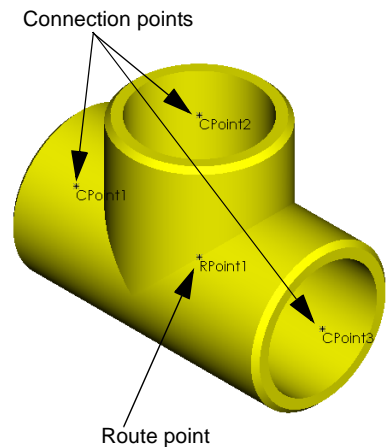
Required Features in Piping Parts

There are two special features of the components used in routing sub-assemblies: connection points and route points.

- ❑ A *connection point* is a point in a fitting where a route begins or ends. A pipe segment can only be generated when at least one end is attached to a connection point. Each fitting part must contain a connection point for each port, positioned where you want the adjoining pipe to begin or end.
- ❑ A *route point* is a point in a fitting that is used to position the fitting at an intersection or endpoint in the 3D sketch.

Some types of fittings use both connection points and route points. For example, the tee shown here has three connection points (one for each port), and one route point (at the intersection of the straight segment and the outlet).

The specific requirements for each type of fitting are discussed in **Routing Components** on page 6-7.



Connection Points



To create a Connection Point:

- 1 Create a sketch point to use for locating the connection point.

NOTE: The location of the connection point defines the *end* of the adjoining pipe.

- If the pipe extends into the fitting, the connection point must be at the correct depth *inside* the fitting.
- If the pipe is welded to the end face of the fitting, the connection point must be coincident with the end face of the fitting.

The distance between the connection points in the route determines the cut lengths of the pipe segments.

- If there is already a sketch of a circle in the correct location, you can use the centerpoint of that circle. For example, if a circular feature is extruded from the face where the pipe ends, you can use the centerpoint of the sketched circle.
 - Otherwise, open a sketch, and add a **Point** . Add dimensions and relations as needed to ensure that the point is concentric with the cylindrical face of the part and positioned at the correct depth.
 - One sketch can contain the sketch points for multiple connection points. In the tee shown earlier in this section, the sketch points for all three connection points and the route point are in a single sketch; the plane of the sketch is coincident with the temporary axes of the three ports.
- 2 Click **Connection Point**  on the Routing toolbar. (If the Routing toolbar is not displayed, click **View, Toolbars, Routing**.)
 - 3 Click the planar face where the pipe exits the fitting, and click the point from Step 1. An arrow is displayed, pointing in the direction that the pipe exits the fitting. If the arrow is pointing in the wrong direction, click the **Reverse direction** check box.
 - 4 Under **Configurable properties**, enter the **Nominal diameter** for the route.
 - 5 Optionally, you can define a **Specification** for each connection point. See the next section for more information about this parameter.
 - 6 Click **OK**.

The **Connection Point** feature contains an **Axis**. When the sketch point was projected onto the face (in Step 3), an axis was created normal to the face at that point. When the fitting is used in a routing sub-assembly, the pipe segment is aligned to this axis.

Connection Point Parameters in the Design Table

You use a design table to control the dimensions in the configurations of the fittings. In addition, you can use the design table to control the parameters of the connection points for each configuration.

See Design Tables in the online help for more information about using design tables to create configurations.

Diameter

When you create a new connection point, you must specify the **Nominal diameter** of the route. When you create additional configurations of a fitting containing a connection point, each configuration needs an appropriate value for the **Nominal diameter**.

NOTE: The **Diameter** parameter cannot be driven by equations. You must specify an explicit value for each connection point in each configuration of the fitting.

If you do not add this parameter to the design table, you must edit the definition of each connection point in each configuration in the part document, and specify the appropriate **Nominal diameter**.

To add this parameter to the design table, use this syntax (in the header cell):

Diameter@CPoint*n*

where **CPoint*n*** is the name of the connection point. If the part has multiple connection points, create a column in the design table for each connection point.

In the body cell for each configuration, enter the value (the nominal diameter). In an elbow part, the value must be the same for both connection points. In a reducer part, the values are different for the two connection points.

To verify that the correct value is used in each case, double-click the **CPoint** feature in the FeatureManager design tree. The **Diameter** is displayed in the graphics area.

The **Diameter** parameter in the connection points of fittings corresponds to the **NominalDiameter@FilterSketch** dimension in the pipe part (see **Pipe Parts** on page 6-7).

Specification

When you create a new connection point, you can define an optional parameter that identifies the wall thickness of the fitting (the **Specification**). You can either use the default name for this parameter or enter a different **Design table name**. For example, if you use ANSI standards, you could enter **Schedule** as the **Design table name**.

You also specify a **Value**, which can be a dimension or other text, according to the standards you use to identify pipe and fitting thicknesses. You can use the same value in more than one configuration.

When you create additional configurations, each configuration needs to specify the appropriate value for the **Specification** parameter.

If you use the default name for **Specification**, the parameter uses this syntax:

Specification@CPointn

– or –

Schedule@CPointn (if you use **Schedule** as the **Design table name**)

where **CPointn** is the name of the connection point.

In the body cell for each configuration, enter the value of the specification. Consult a mechanical design handbook for tables of standard dimensions and specifications, or use the standards established within your company.

If the connection points in your routing parts include the optional parameter, you can select the **Use specification** check box in the **Fabricated Piping** dialog box. The software looks at the connection point parameters where you begin the route. Then, for the elbow part and any other fittings that you add, the software looks for a configuration with the same parameters.

If a matching configuration is found, that configuration is used; if not, you are presented a list of configurations of the correct diameter, and asked to select a configuration.

If you do not use the **Specification** parameter, or if you do not select the **Use specification** check box, you must select a **Base configuration** for the elbow when you start the route, and you must select the appropriate configuration each time you add a fitting.


The **Specification** parameter in the connection points of fittings corresponds to the **\$PRP@Pipe Identifier** property in the pipe part (see **Pipe Parts** on page 6-7).

Route Points

A **Route Point** is used to position a fitting at an intersection or end point in the 3D sketch.

In a fitting with multiple ports (such as a tee or cross), create a sketch point at the intersection of the axes of the fitting before adding a **Route Point**.

To create a Route Point:

- 1 Click **Route Point**  on the Routing toolbar. (If the Routing toolbar is not displayed, click **View, Toolbars, Routing**.)
- 2 Click a point on the model.
 - In a fitting with multiple ports, click the sketch point at the intersection of the axes.
 - In a flange, click the connection point to place the route point coincident with it, or click another point that is concentric with the cylindrical faces of the part.

Routing Components

Pipe Parts

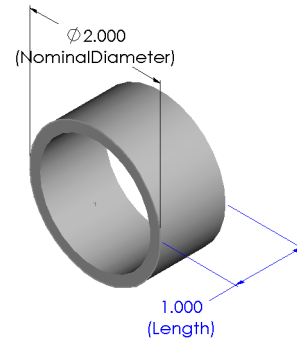
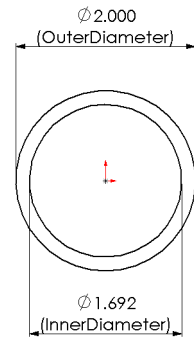
In a pipe part, each type and size of raw stock is represented by a configuration. In the routing sub-assembly, the individual segments are configurations of the pipe part, based on the nominal diameter, the pipe identifier, and the cut length.

IMPORTANT: Be sure to use the exact names specified in the following procedure for sketches, dimensions, and properties of the pipe part.

To create a new pipe part:

- 1 Open a new part, and create a sketch. Name the sketch **PipeSketch**.
- 2 Sketch and dimension two concentric circles. Name the dimensions **OuterDiameter** and **InnerDiameter**.
- 3 Extrude the sketch to any length, then change the name of the **Base-Extrude** feature to **Extrusion**.
- 4 Name the dimension of the extrusion depth **Length**.
- 5 Select one of the planar faces of the pipe, and open a new sketch. Name the sketch **FilterSketch**.
- 6 Sketch another circle, and add a concentric relation to one of the circular model edges. Dimension the circle, and name the dimension **NominalDiameter**. Set the diameter of the circle to the nominal diameter of the pipe.
This dimension corresponds to the connection point parameter called **Diameter** in the fitting parts.
- 7 Close the sketch.
- 8 Insert a design table to create the configurations. In the header row, include the parameters shown in this example.

PipeSketch sketch



Consult a mechanical design handbook or your company standards for tables of standard dimensions.

	A	B	C	D	E
2		OuterDiameter @PipeSketch	InnerDiameter @PipeSketch	NominalDiameter @FilterSketch	\$PRP@ Pipe Identifier
3	1 inch schedule 40	1.315	1.049	1	schedule40
4	1 inch schedule 80	1.315	0.957	1	schedule80
5	2 inch schedule 40	2.375	2.067	2	schedule40
6	2 inch schedule 80	2.375	1.939	2	schedule80

The parameter **\$PRP@Pipe Identifier** is a property that identifies the wall thickness (and other properties, such as material, if desired) of the pipe. This property corresponds to the **Specification** parameter of connection points in fitting parts.

You can include additional parameters as needed, for properties such as weight per unit of length, cost, part number, and so on. For more information about creating and using design tables, see Design Tables in the online help.

- 9 Save the part in the folder specified as the **Library folder path** under **Tools, Options, Routing**.

Elbow Parts

The software inserts an elbow automatically wherever there is a fillet in the 3D sketch. You can also add elbows manually. See **Adding Fittings Manually** on page 6-17.

An elbow part can contain configurations for many different types and sizes of elbow. If you want, the software can create custom elbow parts where needed. See **Routing Options** on page 6-12 for information about this option.

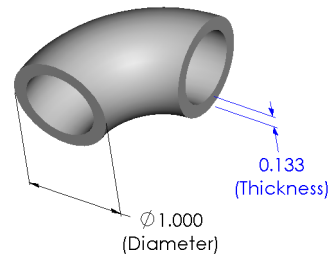
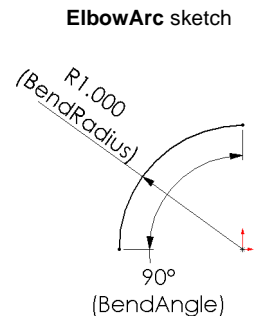
IMPORTANT: Be sure to use the exact names specified in the following procedure for sketches, dimensions, and properties of the elbow part.

To create a simple elbow part:

- 1 Open a new part and create a sketch. Name the sketch **ElbowArc**.
- 2 Sketch and dimension an arc with the radius and angle of a standard elbow. Name the dimensions **BendRadius** and **BendAngle**.

These dimensions correspond to the fillet radius and the angle between the segments in the 3D sketch.

- 3 Add relations as needed to fully define the sketch. For example, in this sketch, there is a coincident relation between the arc centerpoint and the origin, and a horizontal relation between the endpoint and the origin.
- 4 Open a new sketch on a plane that is normal to one end of the arc. Name the sketch **Route**.
- 5 Sketch and dimension a circle. Name the diameter dimension **Diameter**. Set the diameter of the circle to the *outside* diameter value of the elbow. Add a concentric relation between the endpoint of the arc and the centerpoint of the circle.
- 6 Insert a **Sweep** feature using the two sketches.
- 7 Insert a **Shell** feature that removes both end faces of the elbow, with an appropriate **Thickness** for the walls of the part.



- 8 Create **Connection Points** where the pipe exits the elbow at both ends. See **Connection Points** on page 6-4.
- 9 Insert a design table to create the configurations. In the header row, include the parameters shown in this example:

	A	B	C	D	E	F-I
2		Diameter @Route	BendRadius @ElbowArc	BendAngle @ElbowArc	Thickness @ Shell1	
3	1 inch schedule 40	1.315	1.000	90	0.133	...
4	1 inch schedule 80	1.315	1.500	90	0.179	...
5	2 inch schedule 40	2.375	2.000	90	0.154	...
6	2 inch schedule 80	2.375	2.500	90	0.218	...

	A	B-E	F	G	H	I
2			Diameter @CPoint1	Specification @CPoint1	Diameter @CPoint2	Specification @CPoint2
3	1 inch schedule 40	...	1.000	schedule40	1.000	schedule40
4	1 inch schedule 80	...	1.000	schedule80	1.000	schedule80
5	2 inch schedule 40	...	2.000	schedule40	2.000	schedule40
6	2 inch schedule 80	...	2.000	schedule80	2.000	schedule80

The columns for **Specification@CPointn** are optional. The columns for **Diameter@CPointn** are required. See **Connection Point Parameters in the Design Table** on page 6-5.

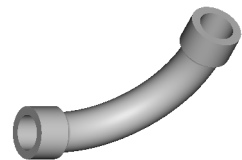
Notice that the values are the same for both **CPoint1** and **CPoint2**. This is required for an elbow part.

You can include additional parameters as needed, for properties such as part number, cost, weight, and so on. For more information about creating and using design tables, see Design Tables in the online help.

- 10 Save the part in the folder specified as the **Library folder path** under **Tools, Options, Routing**.

You can create other types of elbows with additional features.

Create a simple elbow with a **Sweep** feature as described in Steps 1-7. Add features as needed before or after the **Shell** feature. Then add the connection points and the design table.

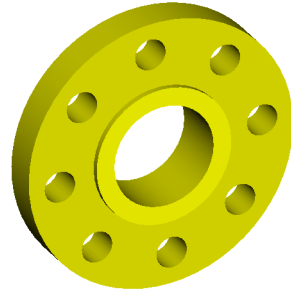


Flange Parts

A flange is often used at the ends of a piping run to connect the pipe to a fixed component, such as a pump or tank. Flanges can also be used to connect long straight sections of pipe.

A flange part must have one **Connection Point** and one **Route Point**. The two points are usually, but not necessarily, coincident. The **Route Point** must be collinear with the axis of the **Connection Point**, and therefore, concentric with a circular edge of the fitting.


The **Route Point** is used to position the flange; the pipe ends at the **Connection Point**.



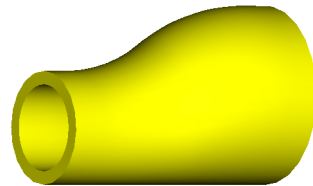
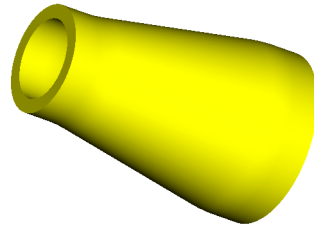
Reducers

A reducer is used to change the diameter of the pipe at a selected location.

A reducer has two **Connection Points** with different values for the **Diameter** parameter.

A reducer also has one **Route Point** (usually in the middle of the part) that positions the fitting at a selected point in the 3D sketch. The point in the 3D sketch can be an intersection of collinear segments, or a point that you insert using the **Split Curve** tool  on the Sketch Tools toolbar.

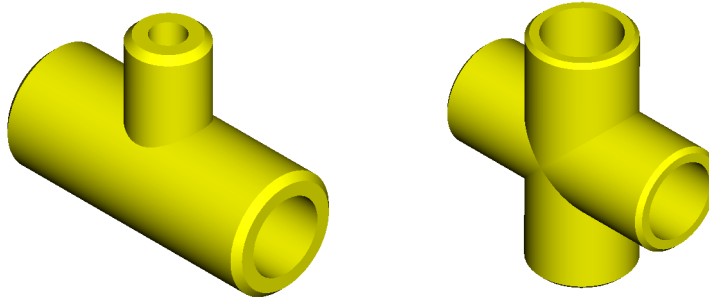
The route point in an offset (eccentric) reducer must be axially aligned with one of the connection points. That axis is used to align the reducer with the 3D sketch segment. You may need to create two versions of these reducers: one whose route point is aligned with the larger port, and one whose route point is aligned with the smaller port.



When you insert the reducer at the end of a sketch segment, select the version that aligns the fitting as needed. To continue the route from the other end of the reducer, right-click the **Connection Point**, and select **Start Route**.

Other Fittings

You can add tees, Ys, crosses, and other multi-port fittings at intersections in the 3D sketch. Fittings with multiple branches must have a **Connection Point** at each port, and a **Route Point** at the intersection of the branches.



When you insert the fitting, the **Route Point** is placed coincident with the intersection in the 3D sketch.

The connection points can have different **Diameters** and **Specifications** if needed.

NOTE: The straight run for a tee must consist of *two separate segments*, not one continuous segment (because two segments of pipe must be created for the straight run).



The straight runs for crosses must also be separate segments.

The number of sketch lines at the intersection can be less than the number of ports in the fitting you want to insert. Insert and align the fitting as desired, then add the remaining sketch segments later. See **Adding Fittings Manually** on page 6-17.

Routing Options

To specify Routing Options:

- 1 Click **Tools, Options, Routing**.
- 2 Specify the **Library folder path** where the routing components are stored. It is recommended that you store all the components that you use in routing assemblies in the same folder. When you create or edit a routing assembly, the software searches in this folder for the components to use.

You can create additional folders under the **Library folder path** if you want to organize your parts in a specific way (by material, by type, and so on).

Some sample routing parts are supplied with the software, in the following folder (unless you selected a different path for the **data** folder when you installed the SolidWorks software):

install_directory\data\Palette Parts\Routing

When you specify a **Library folder path**, that path is automatically included in the **Folders** list for **Palette parts** (under **Tools, Options, External References**). This allows you to add fittings by dragging them from the Feature Palette window.

See External References in the online help for more information about the **Folders** list.

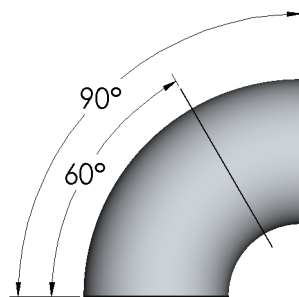
- 3 Select from these options:
 - **Create custom fittings.** When selected, the software automatically creates custom configurations of the default elbow fitting when needed. This is done only in cases where a standard elbow configuration can be cut down to create a custom elbow.

To create a custom configuration, the **BendAngle** (the angle *through which* the material is bent) in the custom elbow must be *less than* the **BendAngle** in the standard elbow.

For example, an elbow with a **BendAngle** of 90° can be cut down to create a 60° elbow.

The software *does not* create custom elbows for variations in the **BendRadius** or **Diameter**.

If your company standards do not permit cut-down elbows, clear the **Create custom fittings** check box. You may need to modify the 3D sketch to use only fillets with standard bend angles.



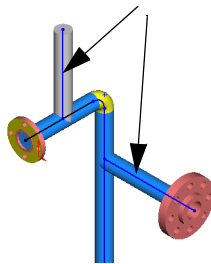
- **Automatically create sketch fillets.** Select this check box if you want the software to automatically add fillets at intersections as you sketch. The fillet radius is based on the **BendRadius@ElbowArc** dimension in the elbow part that you select for the route.
- **Create pipes on open line segments.** Select this check box if you want a pipe to be generated for 3D sketch segments that are connected to a fitting at *only one* end.

For example, if you have not yet added a flange at the end of the route, the last segment of the sketch (beyond the last fitting in the run) is an open segment. If this check box is cleared, no pipe is generated on the open segment.

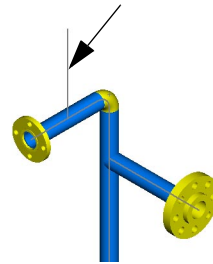
Pipes are never generated for segments that are not connected to a fitting at either end. For example, when you create t-junction in the sketch, the branch is connected to the straight run segment at one end, but not to a fitting. You must add a fitting at the other end of the branch, or add a fitting at the intersection, to create a pipe for the branch.

In this example, a pipe is generated for the branch with a flange fitting, but no pipe is generated for the branch without a fitting.

Branch segments
in the 3Dsketch



No pipe on the
dangling segment



4 Click **OK**.

Creating a Route

There are two ways to start a route.

- ❑ You can begin a route immediately upon inserting a flange or other fitting into a saved assembly document. This is a good method to use if you can mate the flange to another component of the top-level assembly using smart mates. The newly inserted flange becomes the first component of a new routing sub-assembly
- ❑ You can insert a flange, then start the route from a connection point when you are ready. This is the method you use if you want to add mates in the traditional way.

To create a route:

- 1 Click **Tools, Options, Routing**, and make sure that the options are set correctly for the route you want to create. See **Routing Options** on page 6-12.
- 2 Create and save a new assembly document.
- 3 Insert the components that you want to connect, and add mates to position the components as needed. Include any components that the pipe needs to go around as well.
- 4 Drag a flange into the assembly from the Feature Palette window, from an open part window, or from Windows Explorer. Select the configuration to use, and click **OK**.

The **Fabricated Piping** dialog box is displayed.

- 5 In the **Fabricated Piping** dialog box, specify the properties of the route:
 - Under **Start or edit**, specify the **Assembly file name** for the routing sub-assembly document.
 - Under **Connection point properties**, the **Name**, **Nominal diameter**, and **Specification** (if this optional parameter exists in the selected point) of the selected connection point are displayed.

Select the **Use specification** check box if you want software to search in the pipe and fitting documents for configurations with the same **Specification**.

- Under **Pipe fitting**, click **Browse**, and locate **File name** for the pipe part to use. By default, the software looks in the folder specified as the **Library folder path** (under **Tools, Options, Routing**).

If you selected **Use specification**, and the pipe part contains a configuration whose nominal diameter and specification match the selected connection point, the appropriate **Base configuration** is listed.

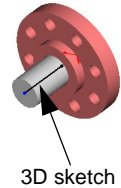
Otherwise, the **Base configuration** list contains the names of any configurations that match the nominal diameter of the connection point. Clear the **Use specification** check box, and select a configuration from the list. The pipe part must contain at least one configuration with the appropriate nominal diameter.

The **Wall thickness** of the pipe is displayed for reference only.


- Select an **Elbow fitting** part in the same way as you selected the pipe part. The default **Bend radius** of the elbow is displayed for reference only.
- To save the current specifications and use them as the defaults in any new assemblies that you create, click the **Save settings** check box.
- Click **Options** if you want to override the default settings (from **Tools, Options, Routing**) for the current route. See **Routing Options** on page 6-12.

- 6 Click **OK**. Switch to **Isometric** view if necessary.

A new routing sub-assembly is created as a top-level assembly component. A 3D sketch for the route is started



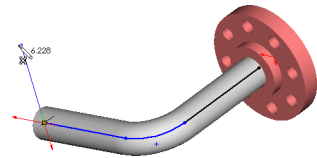
- 7 If the flange is mated correctly by smart mates, skip to Step 11.


If the flange is not mated the way you want it to be (if you did not add smart mates while inserting the flange), exit the 3D sketch now. Right-click anywhere in the graphics area and select **Exit Sketch**, or click the **3D Sketch** tool  on the Sketch toolbar.

- 8 Save the pipe part with a new name. The configurations that are created for each cut length in the route are stored in this file.

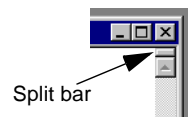
- 9 Mate the flange as needed.

- 10 Open the 3D sketch for editing. Right-click the **Route** feature in the route sub-assembly, and select **Edit Route**.




- 11 Sketch the path of the route using the **Line** tool  on the Sketch Tools toolbar. Press **Tab** to change from one sketch plane to another.

To help you visualize the sketch in 3D space, you may find it useful to split the graphics window. Try displaying an isometric view in shaded mode in one pane, and a front or top view in wireframe mode in the other pane. To split the window, drag the split bars at the corners of the window.



Add dimensions and relations as needed. For example, to create a t-junction between segments on two different sketch planes, add a coincident relation between the straight run segment and the endpoint of the branch segment. For more information, see **3D Sketch** on page 2-9.

If you selected the **Automatically create sketch fillets** option, fillets are added automatically at intersections as you sketch. If you want to add sketch fillets manually (for example, when you need a bend radius that is different from the default), use the **Fillet** tool  on the Sketch Tools toolbar.

To display the virtual sharp intersections in the sketch, click **Tools, Options, General**, and select the **Display virtual sharps** check box.

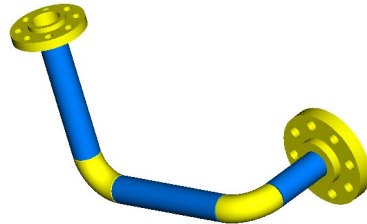
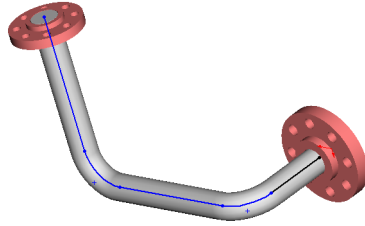
- 12 Add fittings as needed. See **Adding Fittings Manually** on page 6-17.
- 13 Exit the sketch. (See Step 7.)
- 14 If you did not already name the pipe file in Step 8, name it now. Each unique cut length is a configuration of the new pipe part.

If any custom elbow fittings are required, each fitting is saved as a separate new part. You are asked to specify a name for each custom fitting.

For more information about creating custom elbow fittings, see **Routing Options** on page 6-12.

The pipe segments and the elbows are added to the routing sub-assembly.

- 15 To return to editing the top-level assembly, right-click and select **Edit Assembly**.



Editing a Route

Editing the 3D Sketch

To edit the 3D sketch and modify the route:

Right-click the **Route** feature and select **Edit Route**.

You can also right-click the **3DSketch** feature in the route sub-assembly in the FeatureManager design tree, and select **Edit Sketch**.


As you drag segments in the sketch, the pipe updates to reflect the changes. When you exit the 3D sketch, the sub-assembly rebuilds.

Adding Fittings Manually

Elbows are added automatically at fillets in the 3D sketch. You can add other types of fittings, such as flanges, tees, crosses and so on, manually.

To add a fitting manually:

- 1 While editing the 3D sketch, drag a fitting from the Feature Palette window and point to the location in the 3D sketch where you want to place the fitting.

The location can be an endpoint or an intersection, or a point that you inserted in the middle of a segment, using the **Split Curve** tool  on the Sketch Tools toolbar.

- 2 Press **Tab** to cycle through the possible orientations of the fitting. When the appropriate alignment is displayed, drop the fitting.
- 3 Select the desired configuration, and click **OK**.

– or –

- 1 While editing the 3D sketch, right-click a point in the 3D sketch, and select **Add Fitting** from the shortcut menu.
- 2 Select the fitting from the **Select a Fitting** dialog box. Select the appropriate configuration, and click **OK**.

The fitting is positioned with its **Route Point** at the selected vertex. The software attempts to align the fitting component correctly with the sketch lines.

To change the alignment of a fitting:

If a fitting is not aligned correctly with the sketch, you can change the alignment.

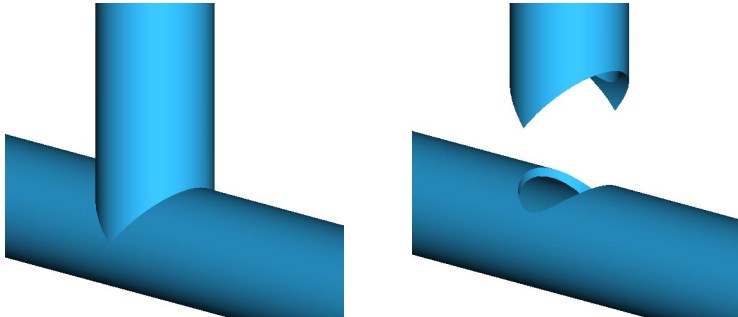
- 1 Exit the 3D sketch.
- 2 Right-click the fitting and select **Change fitting alignment**. Notice the pointer.
- 3 Click the right-mouse button or press **Tab** to cycle through the possible alignments.
- 4 When the desired alignment is displayed, click the left-mouse button or press **Enter**.



Pipe to Pipe Penetration

If cost or space considerations demand, branching fittings such as tees can be eliminated by welding the pipes at the intersection.


In the exploded view shown here, you can see that the branch (the pipe that intersects the straight run) is cut using the profile of the straight run. The straight run is cut using the inner profile of the branch.

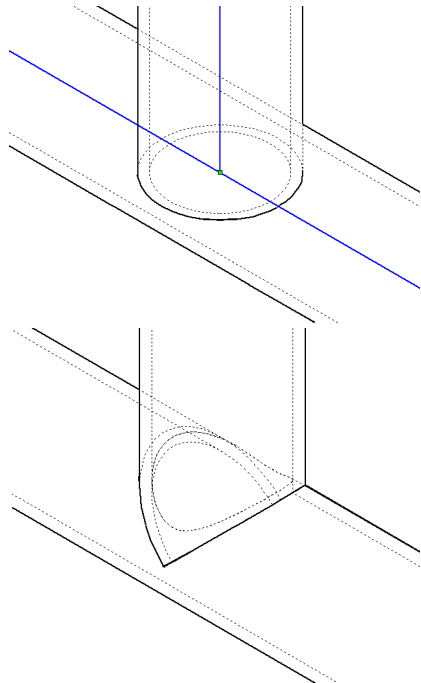


In general, you should perform all editing of the route in the context of the top-level assembly that contains the routing sub-assembly. However, to perform pipe to pipe penetration, you must edit the routing sub-assembly in its own window.

To cut pipes that penetrate each other:

- 1 Right-click the routing sub-assembly in the FeatureManager design tree, and select **Open route_assembly.sldasm**.
- 2 Right-click the **Route** feature, and select **Edit route**.
- 3 Right-click the point where the pipes intersect, (the intersection of the 3D sketch segments), and select **Penetrate**.

The cuts are created as in-context features of the routing sub-assembly. Notice the **Update Sketch**  features at the bottom of the Feature Manager design tree).



NOTE: For pipe to pipe penetration, the straight run must be a *single* segment in the 3D sketch, because the straight run is a single segment of pipe. The endpoint of the branch segment must be coincident with the segment for the straight run. You cannot penetrate at an intersection of three segments.



Reorganizing a Routing Sub-assembly

You can use the assembly restructuring functionality to break down large pipe assemblies into manageable proportions.

- ❑ You can *form a new routing sub-assembly* from selected components in a routing sub-assembly. With the top-level assembly active, select one or more components, then right-click the desired components and select **Form New Sub-assembly Here**. The new sub-assembly is a routing sub-assembly, but it does not contain a route of its own. The new sub-assembly is driven by the route of its parent assembly.

You can create multiple nested sub-assemblies in a routing sub-assembly, with as many levels as you want.

- ❑ You can *move a component of a routing sub-assembly* to a parent or child assembly that uses the same 3D sketch to drive the route. You can either drag and drop the components, or use **Tools, Reorganize Components**. You cannot move a routing component outside of its own branch of the hierarchy. You cannot move components from outside the routing sub-assembly into the routing sub-assembly.
- ❑ You can *dissolve a routing sub-assembly* if its parent is a routing sub-assembly. Right-click the nested sub-assembly, and select **Dissolve Sub-assembly**. You cannot dissolve a routing sub-assembly that contains a route and its 3D sketch; therefore, you cannot dissolve the routing sub-assembly that is the top-level assembly component.

See **Assembly Structure Editing** on page 4-3 for more information about reorganizing assembly components.

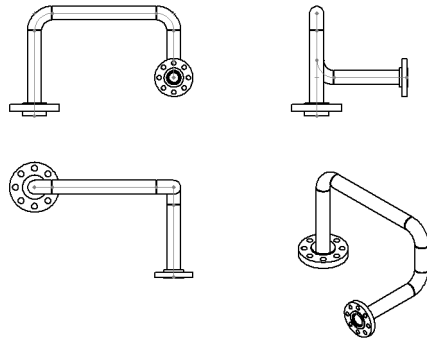
Drawings of Pipe Assemblies

When you create a drawing of a routing sub-assembly, the centerline of the pipe is displayed in the standard orthographic views.

The centerline is not displayed in detail views, section views, broken views, or non-orthographic views (such as isometric views).

If you do not want the route points or connection points to be displayed in the drawing, turn off the display of points in the assembly document (click **View**, and clear the check mark on **Points**).

You can also turn off the display of the 3D sketch. In the assembly document, right-click the **3DSketch** under the routing sub-assembly, and select **Hide** from the shortcut menu.



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