

# SolidWorks® 98Plus

## **What's New**

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# **Module 1**

## **Introduction**





## About this Book

There are many new features available in SolidWorks® 98Plus, spread across the three major applications of modeling, drawings and assemblies. To help you get up to speed quickly, this guide has been created to highlight the new functionality. This manual is for experienced users and assumes you have a good working knowledge of SolidWorks 98.

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

## Using this Book

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

### 1 Load SolidWorks 98Plus.

Load the software from the CD using the Setup script. Be sure to select the option for loading the sample files because the training files are located in the `Samples` directory. Make sure that SolidWorks 98Plus is correctly installed and working.

### 2 Copy the tutorial files to a local directory.

Create a new directory on your system for storing the training files. Copy the files into that directory.

### 3 Follow the instructions.

It is best to go through the document from the beginning, opening the proper file (when requested) that goes with each example. You may however wish to skip to a particular feature of interest, using the Table of Contents to guide you.

**Note:** Not all the new functions require tutorial examples. Many of the enhancements only require one or two sentences of explanation.

## A Word About Files

Since some tutorial files are used with more than one example, it is recommended that you *do not* save changes to these files.

## Format of this Manual

This manual is organized into five modules. They are:

1. Introduction
2. General Information
3. Modeling Enhancements
4. Drawing Enhancements
5. Assembly Enhancements

Following the Introduction, each remaining module contains relatively simple, straightforward examples that illustrate each new feature or enhancement. Read all directions and follow any instructions given.

## Conventions Used in this Book

This manual uses the following typographic conventions:

Convention	Meaning
<b>Bold Sans Serif</b>	SolidWorks commands and options appear in this style. For example, <b>Insert, Boss</b> means choose the <b>Boss</b> option from the <b>Insert</b> menu.
Typewriter	Feature names and file names appear in this style. For example, <code>Sketch1</code> .

## WindowsNT™ 4.0

All the screen shots in this tutorial were made using SolidWorks 98Plus running on WindowsNT™ 4.0. If you are running on a different version of Windows, you may notice differences in the appearance of the menus and windows.

## Last Minute Changes

All attempts have been made 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 late-breaking information that could not be included in this printed book.

## **Module 2**

# **General Information**



## Introduction

Changes that have been made to dialog boxes, menus, views, the FeatureManager™ design tree, and general functions like dimensioning are summarized here. Most of these changes are self-evident and don't require examples or exercises. These changes were made to add functionality or to streamline the use of SolidWorks.

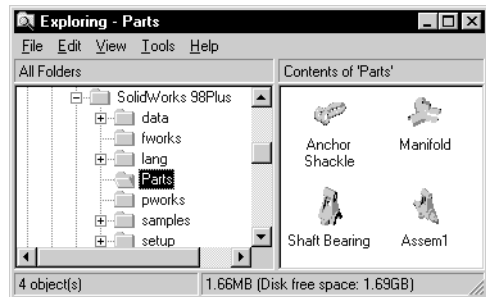
## Options Dialog

The following changes have been made to the **Options** dialog.

### General Tab

The option **Show thumbnail graphics in Windows Explorer** has been added. When this is enabled, thumbnail images of SolidWorks parts and assemblies will appear in the Explorer as well as in the **File, Open** and **File, Save As** dialogs.

**Use shaded face highlighting** turns on or off face highlighting in shaded models. Refer to page 2-6 for more information.



### Edges Tab

You can now display the visible edges of a part while in shaded mode. Under the heading **Shaded Display**, you have the following options:

- **No Edges** — normal shaded display
- **HLR Edges** — superimposes only the visible edges on the shaded display
- **Wire Edges** — superimposes all edges on the shaded display

### Detailing Tab

A new dimensioning standard, **GOST** (Russian), is now supported. For more information on the **GOST** standard, refer to *Module 4: Drawing Enhancements*.

Two font buttons have been added. One is **Section Font...** and the other is **Detail Font...**. These are for the section line and the detail circle text fonts, respectively. For existing drawings (pre-SolidWorks98Plus), these fonts will be set to the dimension text font by default, which is what these items used to follow.

### Line Font Tab

**Cosmetic Threads** can now be assigned a specific line font via the **Type of Edge** pulldown menu. Just like any other default setting, you can change the **Line Style** and **Line Weight** to suit your needs.

## Performance Tab

The performance of large assemblies has been enhanced by a new state for assembly components called “lightweight”. A lightweight component is one which has only a subset of its information loaded into memory. In particular, its shaded display information and certain modeling information are loaded. Because less information is loaded into memory, the performance of assemblies that have lightweight components is faster than with resolved (fully loaded) components.

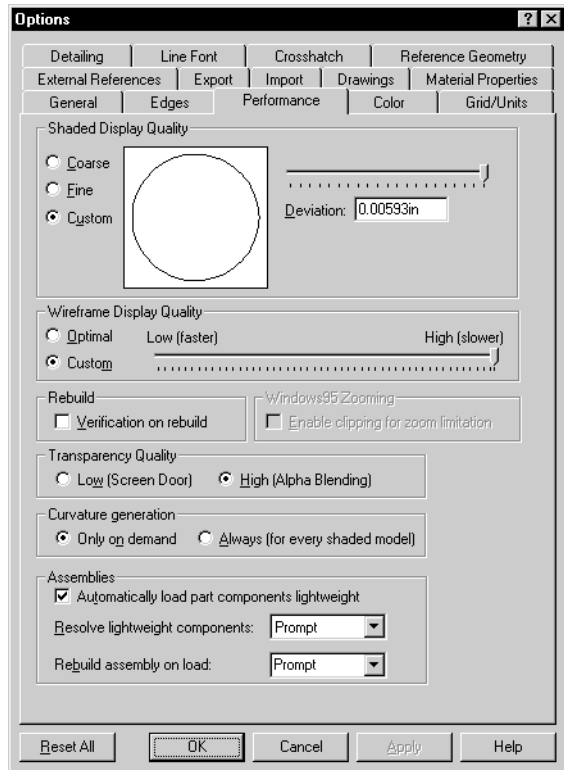
The new option **Automatically load part components lightweight** allows you to automatically load assemblies that were saved in shaded mode in SolidWorks 98 with all part components set to lightweight. This allows quick retrieval of large assemblies and takes advantages of the benefits listed above.

Operations such as solving mates and in-context features that reference a lightweight component will still work as if the components were resolved. However, some operations require fully loading the affected components. Examples of these are non-shaded display modes, mass properties calculations, interference detection, adding new mates, and solving assembly equations.

The behavior of the system when it needs to resolve (fully load) a component in order to perform an operation, is controlled by the option **Resolve lightweight components**. Choose one of the options **Prompt** or **Always** from the pulldown. For more information, refer to *Module 5: Assembly Enhancements*.

The option **Rebuild assembly on load** has been added to control how assemblies are loaded. Choose one of the three options **Always**, **Never** or **Prompt** from the pulldown menu.

- **Always** — Assemblies will be rebuilt without prompting when opened.
- **Never** — Opens the assembly without rebuilding or asking.
- **Prompt** — Asks you whether to rebuild or not when opened.



## Drawings Tab

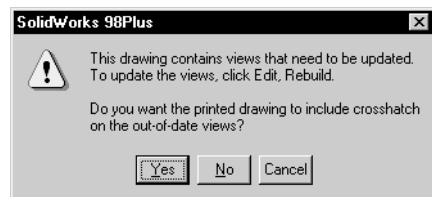
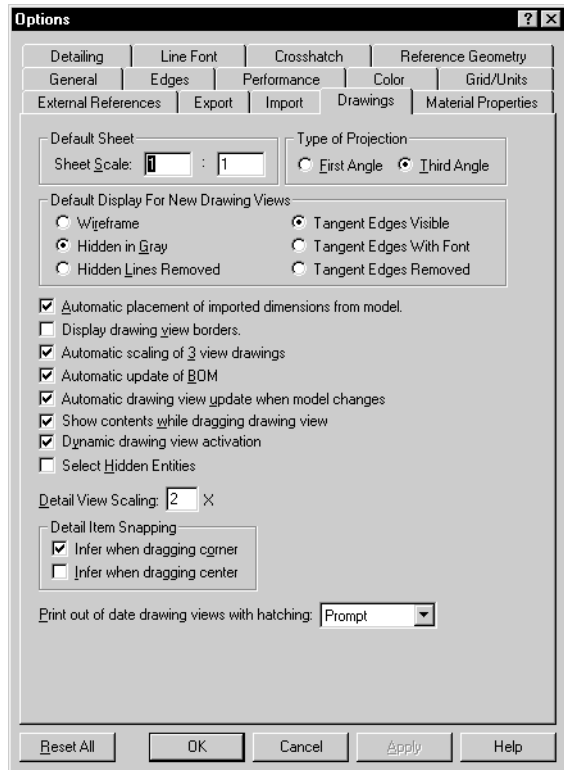
Using the new option **Dynamic drawing view activation**, you no longer need to activate a view in order to attach items such as section lines or detail circles. With this option enabled, the system will dynamically activate the view or sheet, depending on the location of the cursor.

In SolidWorks 98Plus, tangent edges that are removed from display will, by default, not be selectable. This is a change in behavior compared to SolidWorks 98. Use the option **Select Hidden Entities** in order to be able to select any hidden edge.

The **Print out of date drawing views with hatching** option has a drop list with three choices: **Prompt**, **Always**, and **Never**. The default condition is **Prompt**.

If you print a drawing, or use **Print Preview**, with out-of-date views and **Auto view regeneration** is **off**, you will be asked if the drawing views should have the hatching displayed on the print. If the **Always** setting is set, the print will always show the hatching on the out-of-date drawing views and the question will never be asked. If the **Never** setting is set, the print will *not* show have hatching on the out-of-date views *and* the you will never be prompted.

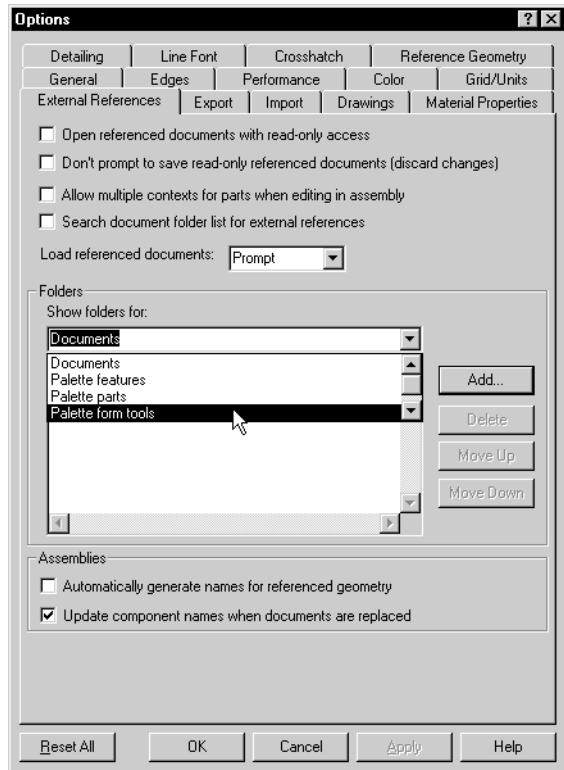
The message you will see is:



## External References Tab

The options listed under **Show folders for** now includes a setting **Palette form tools**. This controls the search path for a library of sheet metal form features such as louvers, lances, and dimples. For more information about form features for sheet metal, refer to *Module 3: Modeling Enhancements*.

The option **Load referenced documents** has been added to control whether externally referenced files, such as base parts or in-context features, are loaded into memory when a file referencing them is opened. Choose one of the three options **Always**, **Never** or **Prompt** from the pulldown menu.



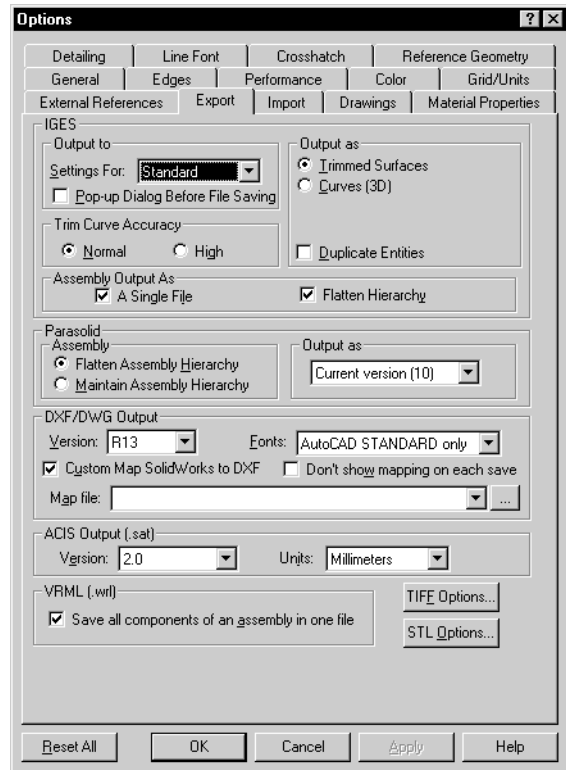


## Export Tab

Under the **IGES** heading, in the section labeled **Assembly Output As**, the option **A single file** has been changed. If it is enabled, the assembly is output as a single file. If it is disabled, a separate file for each part is created. This combines the functions of two buttons in SolidWorks 98 into a single checkbox.

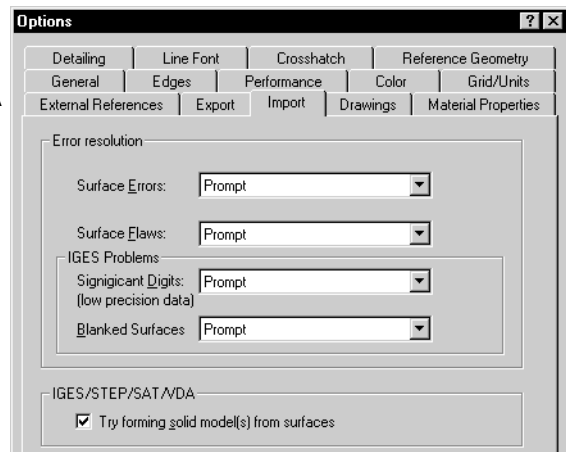
A new option, **Flatten Hierachy**, has been added. This gives you the choice of maintaining the assembly structure (i.e. levels of subassemblies) when writing an IGES file, or flattening the structure so that there are no subassemblies.

**DXF** mapping now supports a **Custom Symbols** entity type.



## Import Tab

The **Import** tab contains a new section for **IGES/STEP/SAT/VDA** data. The checkbox is labelled **Try forming solid model from surfaces**. If this option is enabled, the system will attempt to create a solid when importing surfaces.



## Highlighting

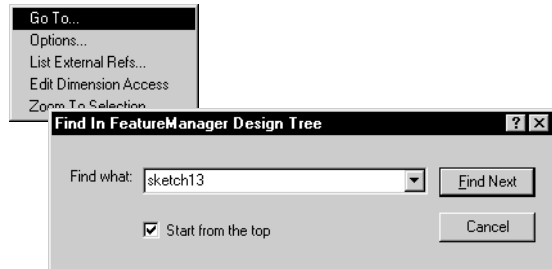
When you select a face of a shaded model, it gets highlighted in the highlight color (green by default). You can change the highlight color through **Tools, Options, Color**. You can turn this option off through **Tools, Options, General**.

## FeatureManager Design Tree

Now the FeatureManager design tree can be searched for a feature name or some portion of it. When the string is found, it is scrolled into view. The same can also be accomplished with a graphic selection.

### Go To Text Search

The **Go To...** option, accessed from the right-mouse menu over the top level component, allows you to search for feature names. In the **Find in FeatureManager Design Tree** dialog, type a full or partial search string in the **Find what** box. Previous search strings will be available through the pulldown portion of the menu. Searching is *not* case sensitive.

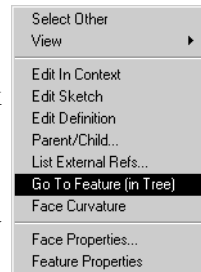


Click **Find Next** to find the next occurrence as searched from the top down if **Start from the top** is checked. If **Start from the top** is not checked, the tool will search from the last position down. When a string is found it is scrolled, if required, into the viewed portion of the FeatureManager window. It is not highlighted. If the string is contained one or more folders, for example, a sketch name that is contained in a feature, the folders are opened to show it.

### Go To Feature — Graphic Search

**Go To Feature (in tree)** is used to locate a feature in the FeatureManager by means of a graphic selection of a face. Click on a face and choose **Go To Feature (in tree)** from the right-mouse menu.

The selected feature will be scrolled, if required, into the viewed portion of the FeatureManager window. Like the previous find function, searching will expand folders to show the feature.

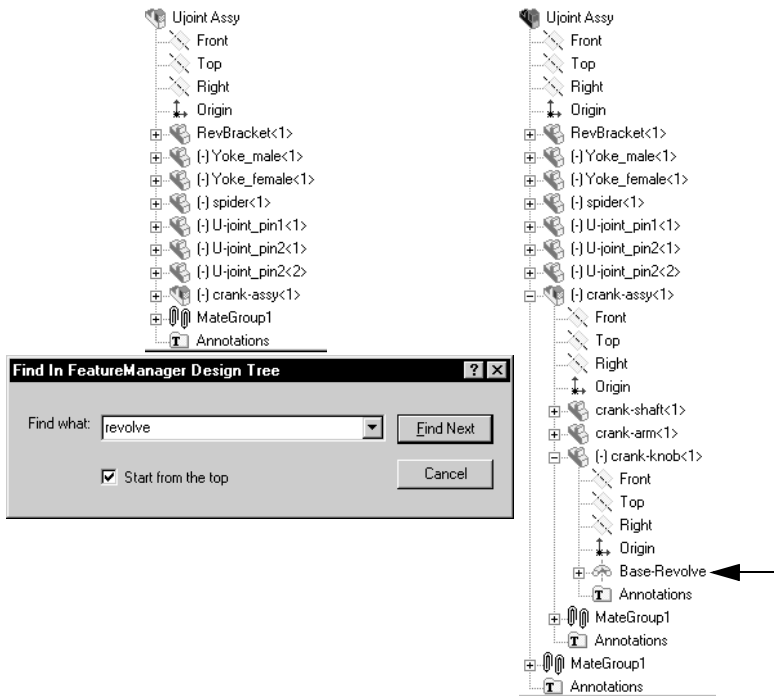


### Window Size

When you resize the FeatureManager window, the new width is saved with the document. When you reopen the document, the FeatureManager window will be restored at the width it was at when the file was saved.

## Example

Using the assembly Ujoint Assy as an example, the text string revolve is searched for. The feature Base-Revolve is found in the subassembly crank-assy. Both the subassembly and part folders are expanded to show the feature.



## Feature Names

Feature names can now use the forward slash “/” character although it is not allowed in component names. @ is not allowed in component or feature names.

## Perspective Views

You can now save a **Perspective** view as a named view. This named view can then be inserted into a drawing.

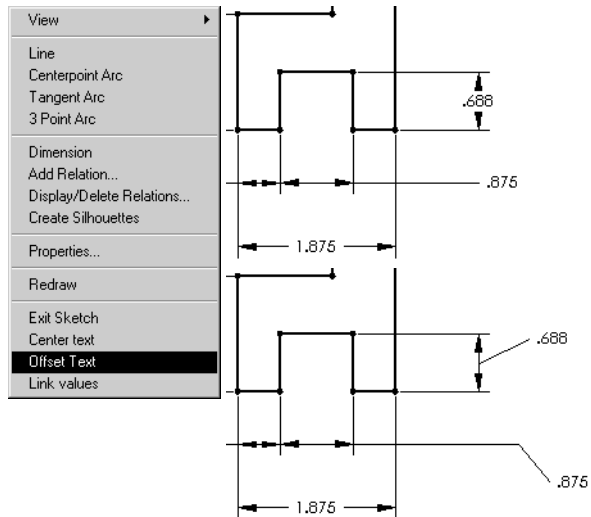
## Multi-selection

Multiple faces of a part can now be selected and changed to the same color, whether in a part or while in **Edit Part** mode within an assembly. **CTRL**-select the faces and use **Face Properties** from the right-mouse menu. You can also change the color of multiple components within an assembly using the same **CTRL**-select method and **Component Properties**. Likewise, you can select multiple components and suppress them using the **Component Properties** option on the right-mouse menu.

## Offset Text

The new right-mouse menu option **Offset Text** creates an offset leader from the dimension to the text. The effect can be toggled off, removing the check mark next to **Offset Text** in the pulldown menu.

This option is available both in sketches and drawings.

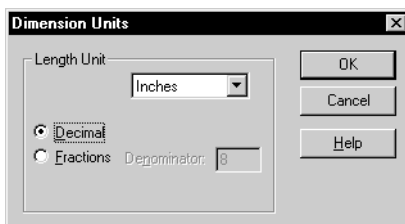
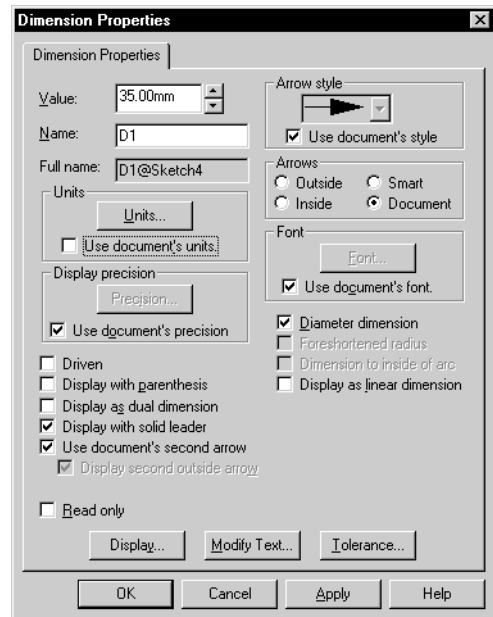


## Dimension Units

A dimension can now have different units than those set in the part, assembly or drawing. This is done on an individual basis using the dimension's **Properties**. For example, this allows you to have both **Inch** and **Feet&Inch** dimensions in the same part or on the same drawing. It also allows you to have some dimensions display in decimals and others in fractions.

Under the **Properties** of a dimension, *deselect* the checkbox **Use document's units**. Click **Units**.

Set the **Length Unit** using the pulldown to the desired units. The current document units appear first.



## Reference Dimensions

When you double-click on a reference dimension, the system opens the **Dimension Properties** dialog. This is the case whether you are in a part, assembly, or drawing.

## Toolbars

A number of changes have been made to toolbars and their buttons. In some cases entire new toolbars have been added to improve access to commonly used commands. In other cases, tools have been redrawn to make their meaning and function clearer and new tools have been created for new operations.

As always, you can customize your user interface using **Tools, Customize**.

## Dependency Editing

This toolbar has been removed. Its tools have been redesigned and placed on the Features toolbar. The new tools are:



**Edit, Suppress**



**Edit, Unsuppress**



**Edit, Unsuppress with Dependents**

## Features

As mentioned above, the Dependency Editing toolbar has been eliminated and its buttons moved to the Features toolbar. In addition, some new buttons have been added. The new functionality represented by these tools is discussed in *Module 3: Modeling Enhancements*.



**Insert, Features, Hole, Simple**



**Insert, Features, Hole, Wizard**



**Insert, Features, Shape**



**Insert, Reference Geometry, Coordinate System**



**Insert, Reference Geometry, Composite Curve**



**Insert, Reference Geometry, Curve Through Reference Points** (Not a new function. It just never had an tool before.)



**Move/Size Features** (Not a new function. Just a redesigned tool.)

## View Orientation Dialog

The **View Orientation** dialog box has been revised to make it smaller. The old text buttons **Add**, **Update**, and **Reset** have been replaced with tools:



**Add**



**Update**

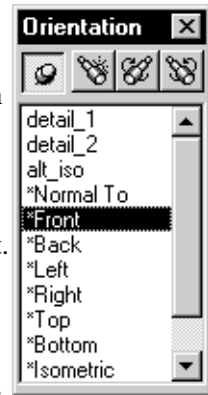


**Reset**

When you add a view, its name still appears at the top of the list.


The specialized view **Undo** button has been redesigned and moved onto the View toolbar . See **View Toolbar** below.

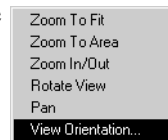
You can also return to previous views (view **Undo**) by using the pulldown menu **View, Modify, Undo View Change** or the keyboard shortcut **CTRL+Shift+Z**.



## Accessing the View Orientation Dialog

Access to the dialog has been improved. In addition to the traditional way, using the **View** pulldown menu, you can:

1. Choose **View Orientation** from the right-mouse menu when the cursor is in the graphics window.
2. Press the **Spacebar**. The spacebar is the default keyboard shortcut.
3. Press the  tool on the View toolbar. See below.



## View Toolbar



The appearance of the tools for **Wireframe**, **Hidden in Gray**, **Hidden Lines Removed**, and **Shaded** has changed subtly. In addition, tools for accessing the **View Orientation** dialog and undoing view orientation changes have been added.

## Standard Views




A new toolbar named **Standard Views** has been added. The buttons are used to change to the standard, predefined views quickly. For most view changes, this eliminates the need to open the **View Orientation** dialog.

Previous views (view **Undo**) can also be accessed through **View, Modify, Undo View Change** or **CTRL+Shift+Z**.

From left to right, the tools are: **Front**, **Back**, **Left**, **Right**, **Top**, **Bottom**, **Isometric**, and **Normal To**.


## Selection Filter Toolbar

The selection filter pulldown menu in SolidWorks 98, with its limited set of filters, has been replaced with a selection filter toolbar. Like most toolbars, you can dock it vertically or horizontally along the edge of the screen, or position it as a separate window inside the graphics area. For illustration purposes and to conserve space in this book, it is shown as a separate window.

When a selection filter is on, the cursor changes like this: .



## Accessing the Toolbar

All toolbars, including this one, can be accessed by placing the cursor in the toolbar area of the SolidWorks window and pressing the right mouse button. Choose **Selection Filters** from the menu. You can also turn the Selection Filters toolbar on and off using the  icon on the Main toolbar.

## How Filters Work

You can control which objects are selectable by turning on some, all, or no selection filters. The three tools located at the top of the filter menu have specialized functions which are explained below. Those with keyboard shortcuts (case sensitive) have them indicated in parentheses.



**Toggle Selection Filters** — This enables/disables whatever set of selection filters you have picked. (**F6**)



**Clear All Filters** — This deselects all filters.



**Select All Filters** — This turns on all the filters.

The remaining buttons represent filters for specific types of geometry or objects. Positioning the cursor over a filter will display a tooltip explaining its function.



**Vertices (v)**



**Sketch Segments**



**Notes and Balloons**



**Edges (e)**



**Midpoints**



**Datum Features**



**Faces (x)**



**Centermarks**



**Weld Symbols**



**Axes**



**Dimensions**



**Datum Targets**



**Planes**



**Finish Symbols**



**Cosmetic Threads**



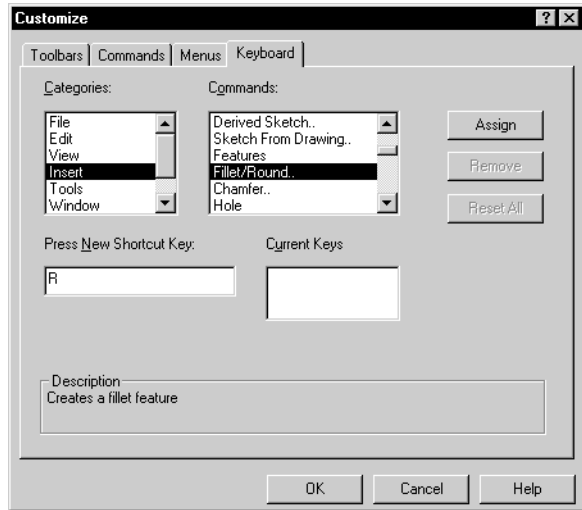
**Points**



**Geometric Tolerance**

## Keyboard Shortcuts

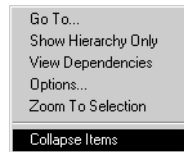
Using **Tools, Customize, Keyboard**, you can create shortcuts of a single character like those already embedded into SolidWorks (i.e. **f** for **Zoom to Fit**).



## Collapse Items

A new option, **Collapse Items**, can be used to close all expanded features in a part, assembly or drawing.

Access **Collapse Items** from the top level icon in the FeatureManager design tree by using the right-mouse menu. All expanded features will be collapsed. Note that if there are no open features, the option will not appear on the menu.



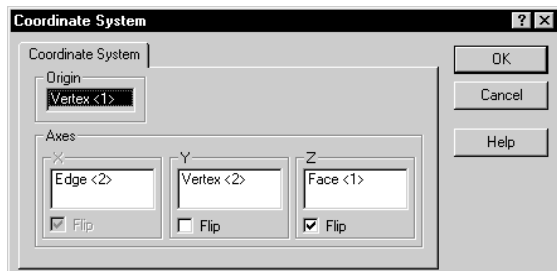
## Coordinate Systems

SolidWorks 98Plus now gives you the ability to create cartesian **Coordinate Systems**. These coordinate systems can be used to map the output of the **Measure, Mass Properties** and **Section Properties** dialogs. They can also be used for data export such as **IGES, SAT, STEP, STL, and VDA** export.

### Creating Coordinate Systems

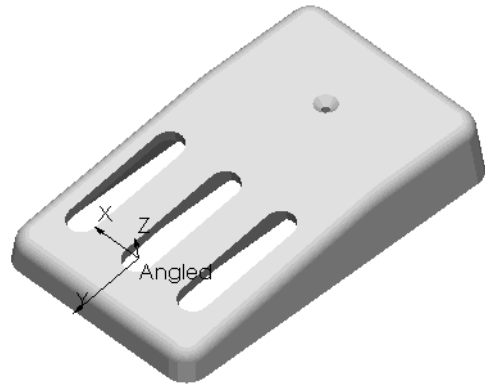
Using either **Insert, Reference Geometry, Coordinate System...** or the tool **Coordinate System** on the Feature toolbar, new cartesian coordinate systems can be added to a part or assembly.

Geometry in the part or assembly such as edges, vertices, edge midpoints, faces, axes, points and sketch geometry can be used to define the **Origin** location and **Axes** directions. The **Flip** button can be used to flip or reverse the direction of the axes.

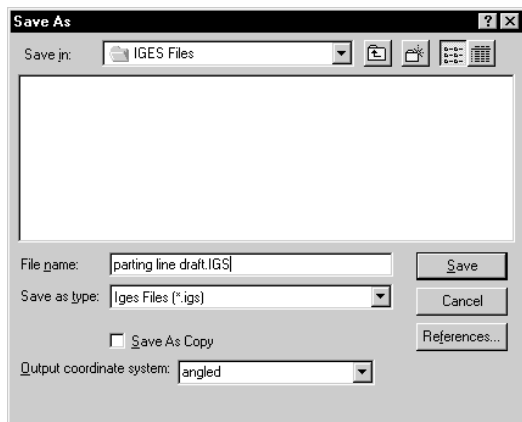




The name of the new coordinate system appears in the FeatureManager design tree and on the screen as **Coordinate System1**. It can be renamed like any other feature. Coordinate systems can be viewed on or off globally using **Tools, Options, Reference Geometry, Coordinate Systems**, or locally using **View, Coordinate Systems**.



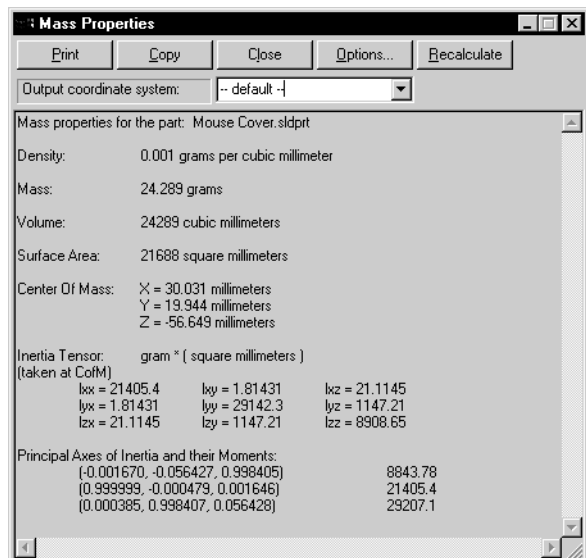
For data export, IGES as an example, the **Save As** dialog has a place for the coordinate system. **Output coordinate system** has a pulldown list that contains all the coordinate systems in the part or assembly. Choose **Default** or any user-defined system.



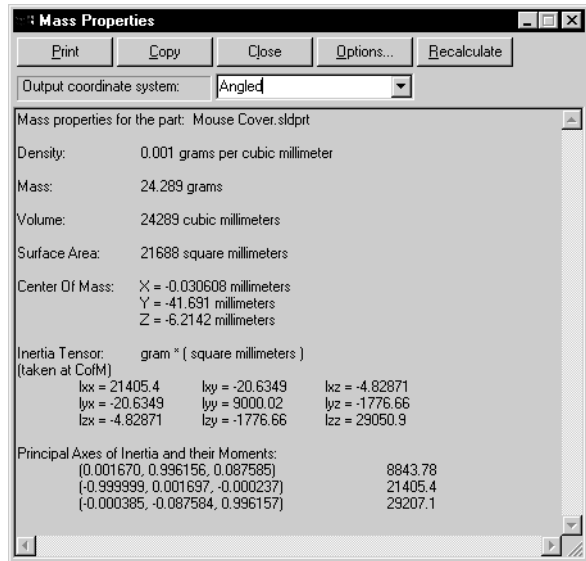
## Using the Coordinate System

Once the coordinate system has been added, it can be used with the **Measure, Mass Properties** and **Section Properties** dialogs.

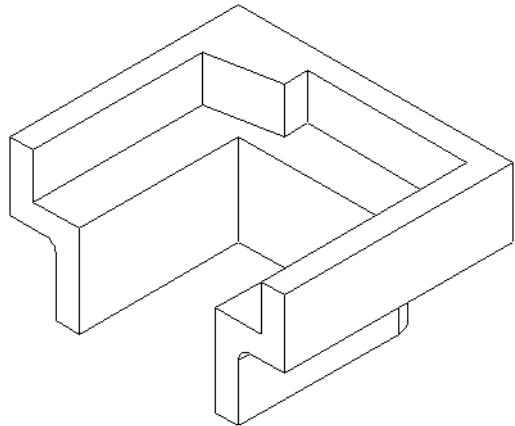
For example, the **Mass Properties** of the part **Mouse Cover** were calculated. Changing the **Output coordinate system** from **-default-** to **Angled** and clicking the **Recalculate** button causes the values to be recomputed with respect to the new coordinate system.



Similar options exist in the **Tools, Measure** and **Section Properties** dialogs.



- 1 **Open the part.**  
Open the existing part `coord_system` in the What ' s New directory.



**2 Coordinate system.**

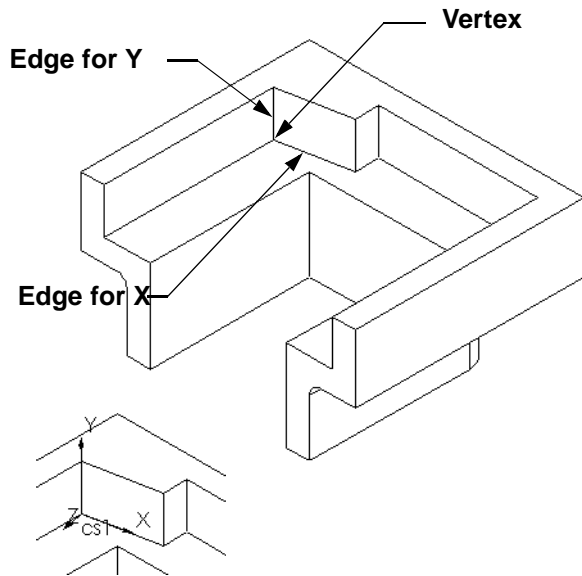
Using a vertex for the **Origin** and two edges for the **X** and **Y Axes**, create a new coordinate system as shown.

Click the **Coordinate System** tool and select the vertex as the **Origin**.

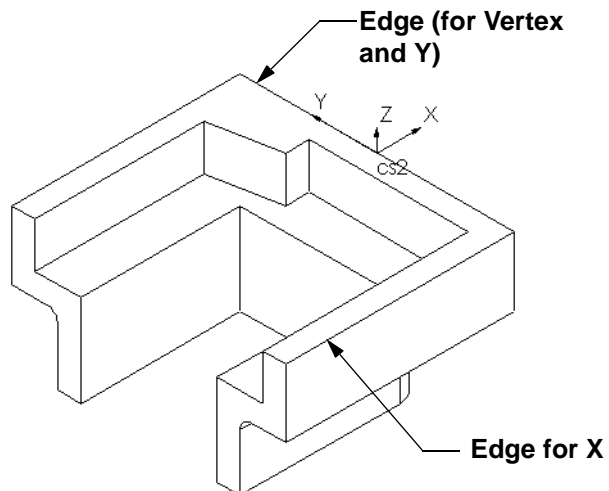
Click in the **X Axis** field and select the indicated edge. Flip to reverse the axis.

Click in the **Y Axis** field and select the indicated edge. Flip to reverse the axis.

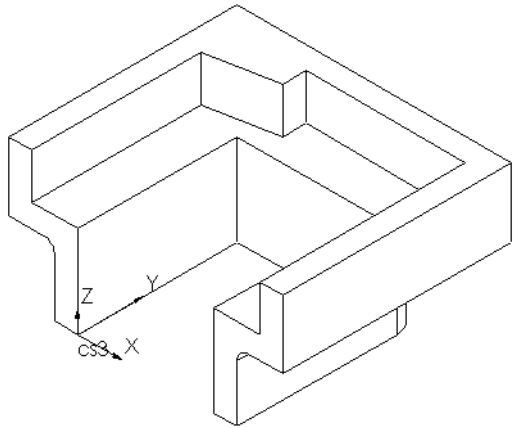
Click **OK** to add the feature. Rename it **cs1**. Show the coordinate system using **View, Coordinate Systems**.

**3 Second coordinate system.**

Create **cs2** using a midpoint as the **Origin** and two edges to define the **X** and **Y Axes**.



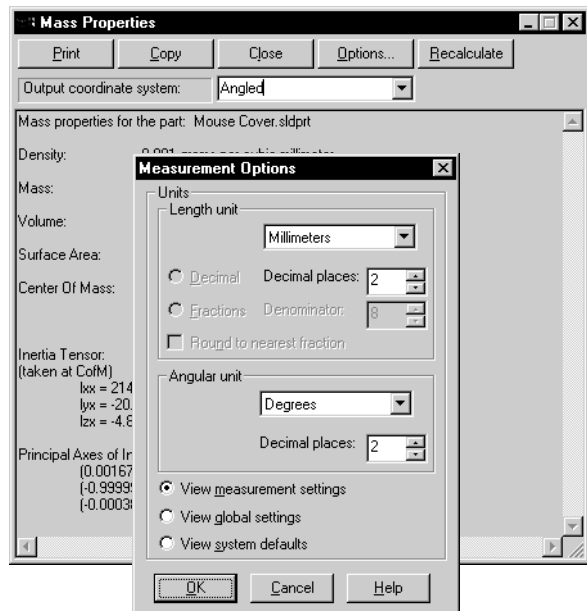
- 4 **Coordinate system.**  
Create cs3 using a vertex and two edges.
- 5 **Use the coordinate systems.**  
The coordinate systems can be used in conjunction with **Mass Properties**, **Section Properties**, **Measure** and **IGES** output. Try these if you wish.
- 6 **Close the part.**



## Measurement Options

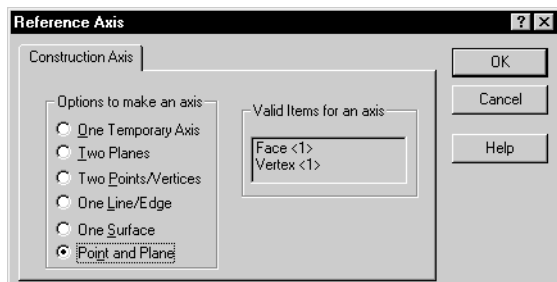
Another change to the dialogs for **Measure**, **Mass Properties** and **Section Properties** is the ability to set the output units. You do this via the **Options** for the dialog. These options determine the **Length**, **Angular** and **Decimal place** settings used by the dialog. The settings can be different than the global (document) or system defaults.

After the **Length**, **Angular** and **Decimal place** settings are made, use **Recalculate** to compute the values using the new settings. Use **View global settings** or **View system defaults** to change back.

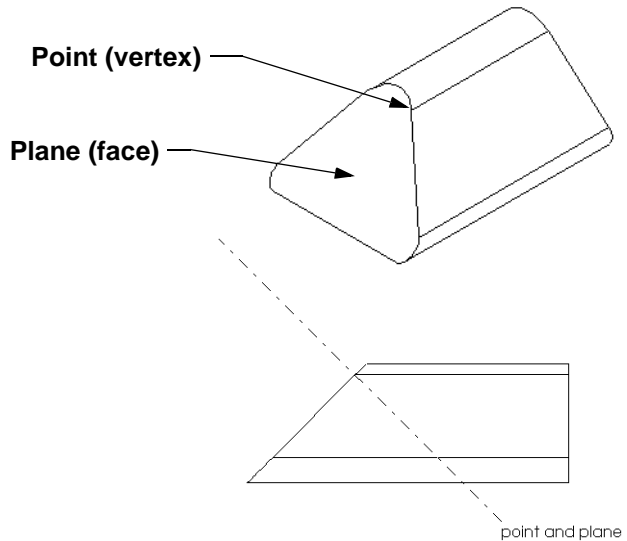


## Reference Axis

The **Reference Axis** tool will now create an axis normal to a plane through a point.



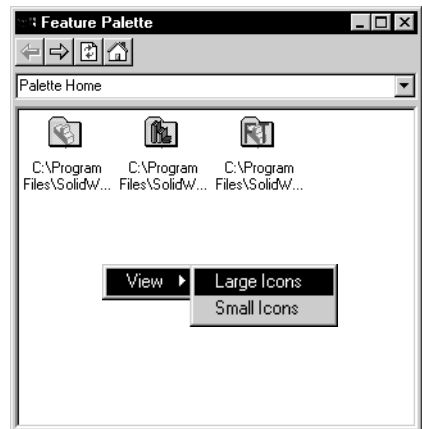
Select a planar face or plane in conjunction with a vertex or endpoint. Use the **Point and Plane** option to create the axis.



## Feature Palette™ Window

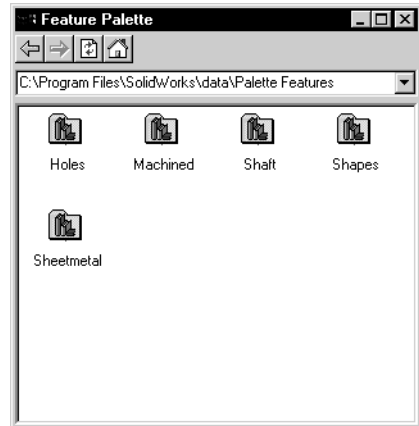
The Feature Palette window has changed in appearance. It is now displayed as a palette page with folders. A small set of navigation tools includes **Back** and **Forward** arrows to move through folder levels, a **Reload** and a **Home** button. The folders display an icon of a part (**Palette Parts**), a library feature (**Palette Features**) or a form feature on them.

The icons can be viewed large or small using the right-mouse menu within the window. Note that only the large option shows pictures of the features.

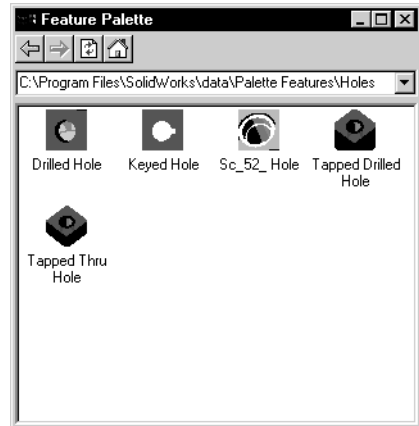


## Using the Palette

Double-click a folder, such as library features, to open it. Sub-folders are listed in the same fashion as the top level ones. The contents of each sub-folder are either parts or library features, as indicated by the icon on the folder. The folders shown all contain palette features. The full pathname of the folder is listed above the icons. The folders are displayed in alphabetical order, left to right and top to bottom.



Double-click another folder, such as Holes, to open it. The contents of this folder are displayed. From this level, the palette features can be dragged and dropped into active parts. Clicking the **Back** arrow twice, or clicking **Home**, will return you to the top level.

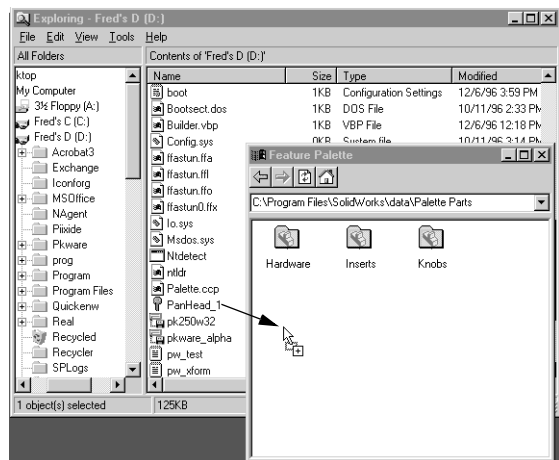


Items can be removed from the Feature Palette by deleting the file from the palette directory.

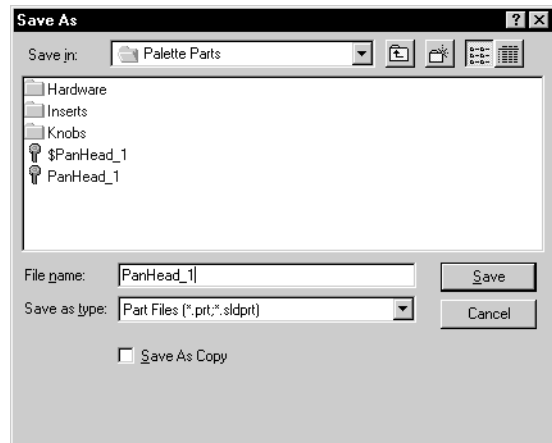
## Adding to the Palette

Parts and library features can be added to the appropriate folders by simply dragging and dropping them into the Feature Palette. You can drag and drop from:

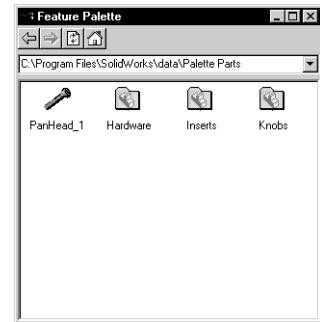
1. Windows® Explorer
2. The top level icon in the FeatureManager of the open document
3. Microsoft Internet Explorer 4.0
4. Within a folder, making a copy of a library part or feature.



Under some circumstances, the standard **Save As** dialog appears so you can rename the file being copied.



The icon appears in the proper directory and visually in the **Feature Palette**.



## Edit Palette Item

Edit Palette Item

A new option, **Edit Palette Item** is available from the right-mouse menu when the cursor is positioned over a file in the Feature Palette window. Using it opens the part or feature for editing directly from the Feature Palette window.

## Palette Feature Names in the FeatureManager

When a palette feature is inserted into a part, the resulting library feature is named the same as the palette feature, rather than Libfeat1.

## Object Snap Changes

The highlighting and appearance of vertices has changed in SolidWorks98 Plus. Snapping to a vertex now appears as a double box on the screen.



## Drag and Drop from Internet Explorer

SolidWorks part files can now be dragged and dropped from Internet Explorer to:

- Feature Palette
- New Part
- Drawing
- Assembly

If the file is dropped into a drawing or an assembly, you will be prompted to name the new file on your local disk.

## Importing and Exporting Data

For ease of reference, the enhancements to importing and exporting data are listed here, in one section.

### IGES

- When importing, surfaces and curves can be filtered by layer.
- When exporting, you can do so relative to user-defined coordinate systems.
- You can import 2D data into a sketch.
- You can export selected components, including subassemblies, from within an assembly.
- You can control whether or not the system tries to knit surfaces during import.

### SAT

- You can export selected faces or reference surfaces.
- You can control whether or not the system tries to knit surfaces during import.

### STEP

- You can export selected faces or reference surfaces.
- You can control whether or not the system tries to knit surfaces during import.
- Importing supports STEP AP214.

### Parasolids (X\_T and X\_B)

- You can import or export face colors for parts and component colors for assemblies.
- You can import and export component names for assemblies.

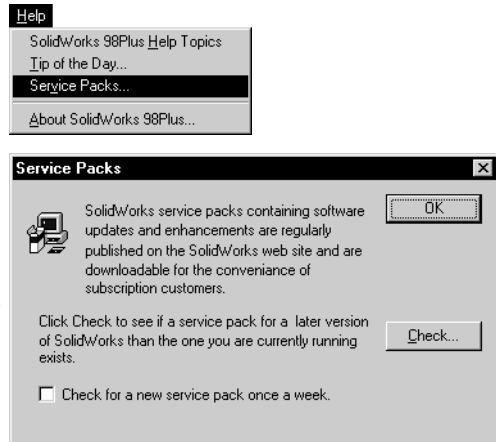
### VDA

- You can export selected faces or reference surfaces.
- You can control whether or not the system tries to knit surfaces during import.



## Service Packs

A new option, **Service Packs**, is available under the **Help** menu. This allows you to conveniently check for new service packs and initiate downloading them. When you pick this option, your default browser will be launched into the support area of the SolidWorks web site. You also have the option to have SolidWorks check for service packs once per week on start-up.



**Note:** Downloading service packs from the SolidWorks web site is a benefit that is only available to customers who have purchased the SolidWorks Subscription Service.

## Converting Files

Any files opened in SolidWorks 98Plus will automatically convert to the updated format and once saved will *not* be backward-compatible.

The first time you open a pre-SolidWorks 98Plus file, it may take longer than normal. Once the file has been opened and saved, the subsequent time needed to open the file will return to normal.

You can use the new **Conversion Wizard** to convert all your files to SolidWorks 98Plus format. This can be found in the SolidWorks

program group on the Windows Start menu. Since this tool will overwrite the existing files, it gives you the option to make a back-up copy of the files. You can also control whether SolidWorks will run in an open window or in background.





## **Module 3**

# **Modeling Enhancements**



## Sketching

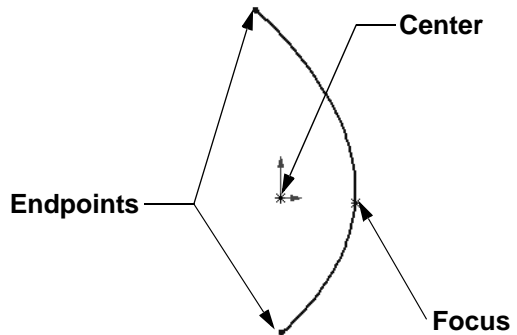
Below is a list of the new features and enhancements that have been added to Sketching in SolidWorks 98Plus. Some of these are illustrated in more depth in examples later in this module. They are listed here in one place with a short description for your convenience and ease of reference.

### Parabola



The sketch option **Parabola** has been added for SolidWorks98Plus. It has four control points: two endpoints, a center and a focus point.


The **Parabola** is created by a method similar to the **Centerpoint Arc**. You press the mouse button to locate the center and drag outward to locate the focus. Release the mouse button.

Then press the mouse button again and drag to position the endpoints. All the control points can then be dragged, dimensioned, or related within the sketch.



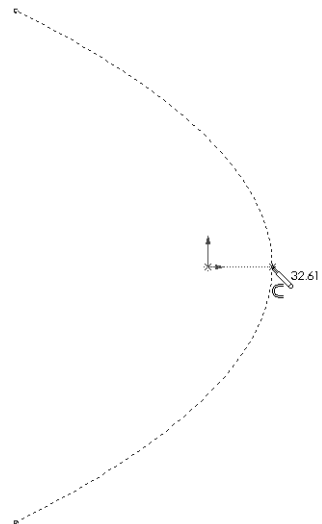
#### 1 Sketch a parabola.

Select the **Parabola** option from **Tools, Sketch Entity** or pick the  button on the Sketch Tools toolbar. The cursor changes to display the parabola symbol .

If the  tool is not on the tool bar you can add it using **Tools, Customize**.

#### 2 Locate the center and the focus.

Position the cursor at the **Centerpoint** location (the **Origin** in this example) and drag away to define the location of the **Focus**. The parabola will be formed in temporary (blue dotted) graphics.

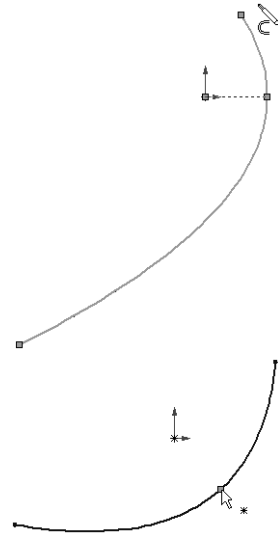


### 3 Drag an endpoint.

Position the cursor near an endpoint, or just near a portion of the curve, and drag and drop to form the perimeter of the parabola. The position that you start dragging from will become the first endpoint of the parabola. The point you stop at will be the second endpoint.

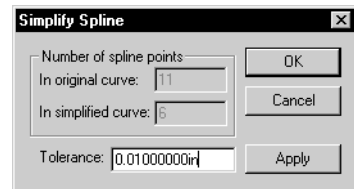
### 4 Drag the control points.

The four control points can be dragged, dimensioned or related to other geometry.



## Simplify Spline

Sketched splines can be simplified by specifying a deviation value. With the cursor over the spline in the sketch, choose **Simplify Spline** from the right-mouse menu.

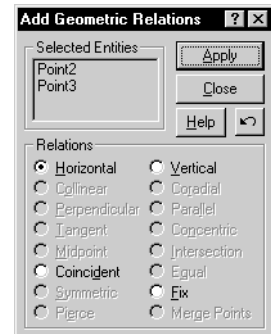
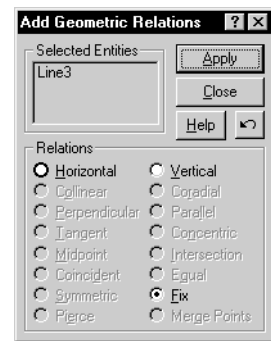
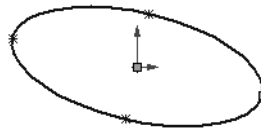


## Add Geometric Relations

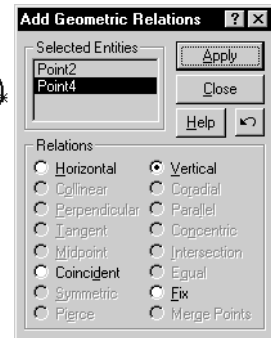
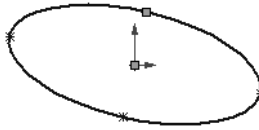
The **Add Geometric Relations** dialog has been improved to provide better “guessing” as to which relation is desired for a given set of selections. The relation pre-selected as the default choice can be overridden by the picking another valid relation.

Relations that already exist between the selected geometry are marked with a flat and darkened ring. The **Horizontal** relation shown at right already exists.

In this example, the centerpoint and the right-hand end of the major axis are selected. The relation **Horizontal** is pre-selected because the points lie closer to the horizontal than the vertical.



In the second example, the centerpoint and the upper end of the minor axis are chosen. **Vertical** is chosen as the default relation based on the objects selected and their positions.



## Revolved Features

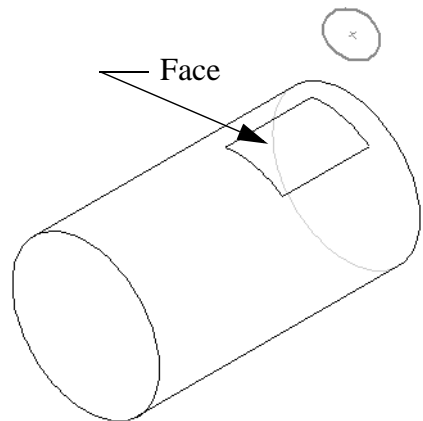
Revolved features, whether a boss or cut, now support multiple sketch profiles. This is consistent with the behavior of extruded features.

## Lofting

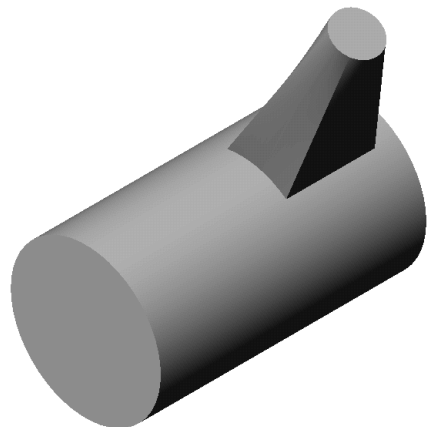
Now 3D profiles can be used in a **Loft** operation. Previously, only sketches could be used.

The 3D profile is created using the **Split Line** option which projects a sketch onto a face. The resulting geometry is a new face.

You must select the new face, not the edges or feature, to use the 3D curve. Select on the face near the desired vertex location.



The resulting **Loft** (a boss in this example) is added to the solid.

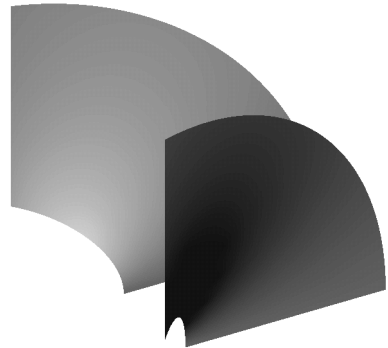


## Lofting Between Surfaces

Imported surfaces and references can also be used as profiles when lofting. This gives you a simple way to incorporate surfaces into solids. Consider the following example:

### 1 Open part.

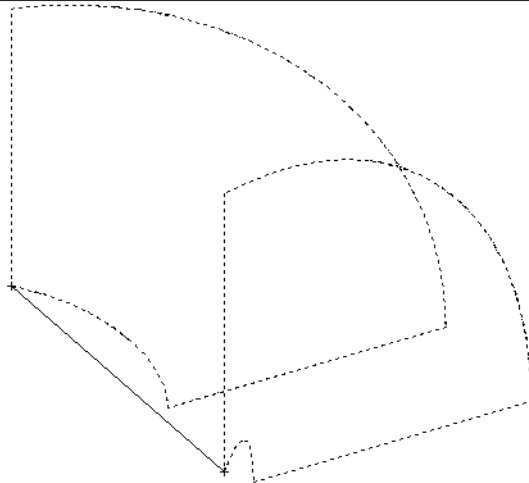
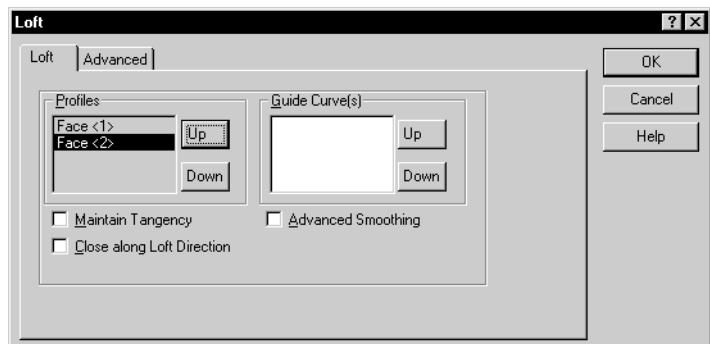
Open the part **Loft Surfaces** in the **What's New** directory. This part contains two imported surfaces.



### 2 Loft a base feature.

Use **Insert**, **Base**, **Loft** to create a solid between the two imported surfaces.

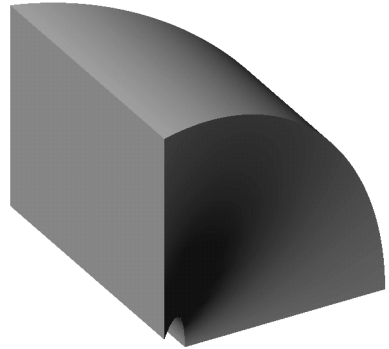
Be sure to pick the faces of the surfaces, not their edges.



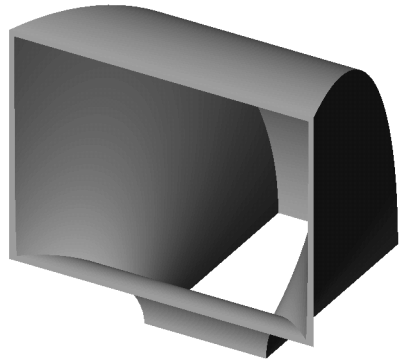


**3 Results.**

The part now consists of a solid base feature which incorporates the two imported surfaces.

**4 Shell the solid.**

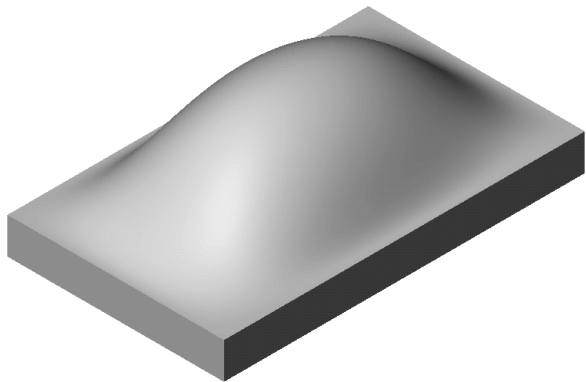
As you can see, once the surfaces have been transformed into a solid feature, other operations can be performed on the model.

**5 Close the part.****Shape Feature**

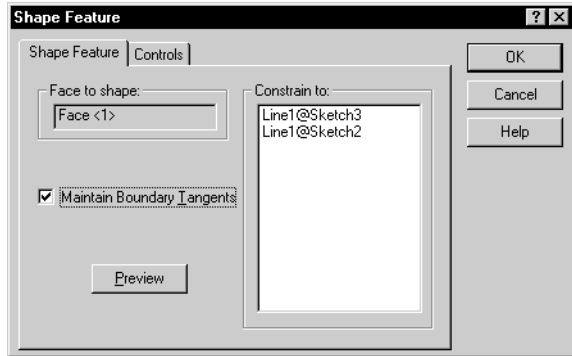
The new **Shape Feature** option can be used to create freeform shapes by stretching and modifying model faces. This option can be accessed from **Insert, Features, Shape Feature** or from the



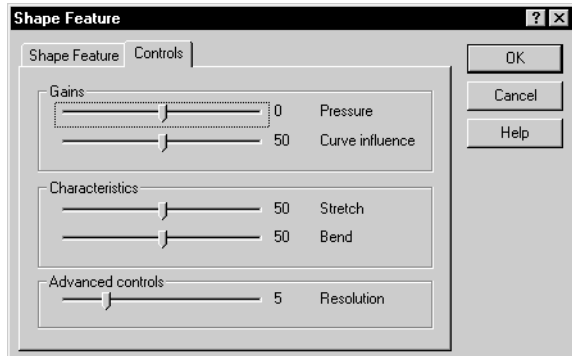
icon on the Features toolbar.



The dialog box contains two tabs, **Shape Feature** and **Controls**. In the **Shape Feature** tab, you identify the **Face to shape** and the sketches, sketch geometry, vertices or edges to **Constrain to**. The **Maintain Boundary Tangents** checkbox will keep the modified face tangent to the original face.



The **Controls** tab has five sliders that are used to influence the shape of the face. Although there are numbers for each category, they are unitless and do not produce dimensions. A click on the slider moves the value 20 units. Use the arrows keys (left and right) to move in one unit increments.



#### ■ **Pressure (Gains)**

**Pressure** is the only slider that starts at zero and can move to positive or negative values. The positive side “inflates” the face while the negative side “deflates” it.

#### ■ **Curve influence (Gains)**

This slider runs from 0 to 100 and influences how closely the face follows the **Constrain to** geometry. The higher the value the more influence the geometry has on the shape.

#### ■ **Stretch (Characteristics)**

The **Stretch** slider can be any value from 0 to 100. The higher the value, the more easily the face stretches.

#### ■ **Bend (Characteristics)**

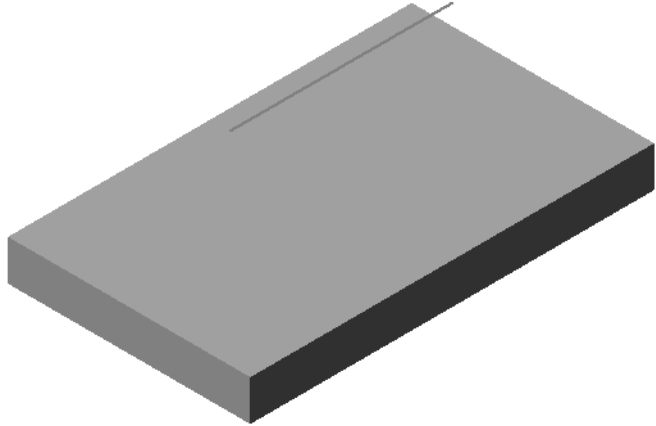
The **Bend** slider (0-100) determines how much resistance the face offers to bending. A high value allows the face to bend easily. A low value makes it stiff.

#### ■ **Resolution (Advanced controls)**

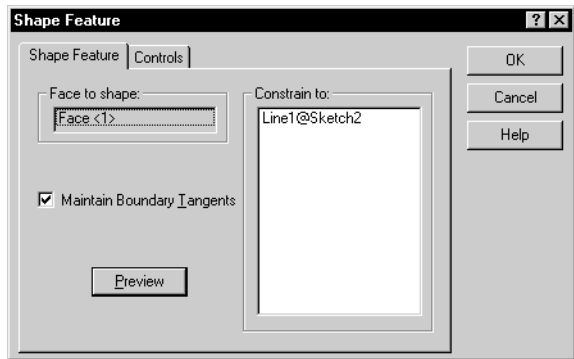
The setting for the **Resolution** determines the size of the “mesh” on the face. More than this, it contributes to the shape by providing more or less sampling points. Higher values will take longer to rebuild.

**1 Open the part.**

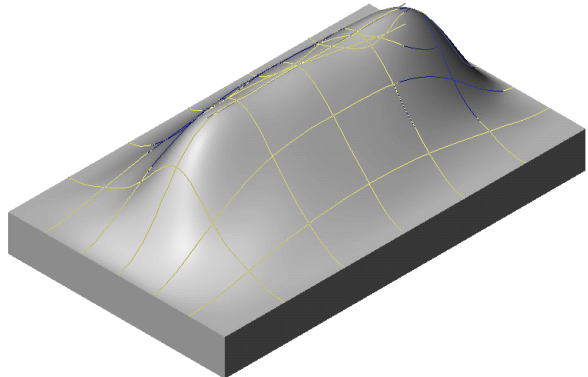
Open the existing part `shape_feature` in the What's New directory. The part is a plate with a visible sketch.

**2 Selections.**

Click the **Shape Feature** icon. Select the top face of the model as the **Face to shape**. Click in the **Constrain to** list and select the line. You can select an entire sketch or just individual geometry.

**3 Click Preview.**

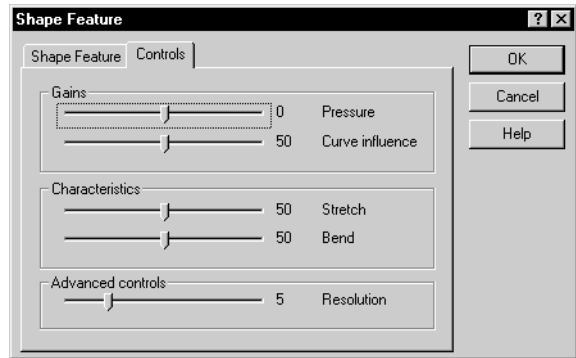
The system performs the necessary calculations and displays the deformed surface. The mesh-like graphics represent the advanced control **Resolution**.



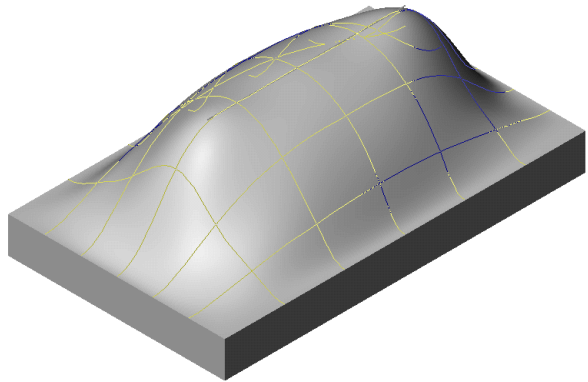
**4 Controls.**


Click the **Controls** tab to see the sliders. By default, the sliders are set to mid position except for **Resolution**, which is set low.

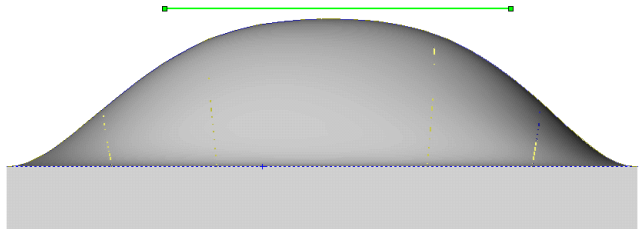
Changes to the sliders will cause changes to the model. When you finish moving a slider (release the mouse button), the model will update.


**5 Pressure.**

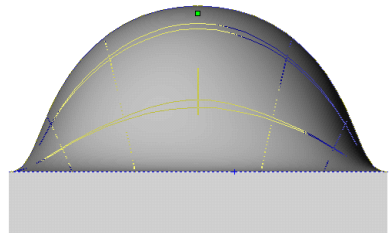
Drag the **Pressure** slider to the right, “inflating” the face slightly. Stop at **20**.

**6 Curve influence.**

Change to the **Right** view using the **Standard Views** button . Set the **Curve influence** to **24**, loosening the conformance to the selected line.

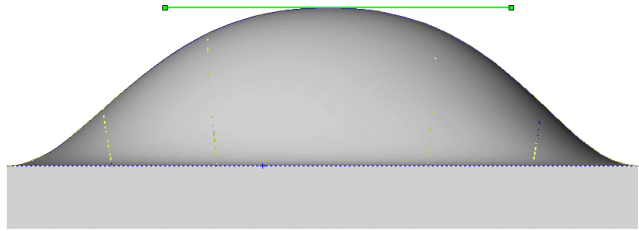
**7 Resistance to stretching.**

Change to the **Front** view using the **Standard Views** button . Set the **Stretch** value to **75**, decreasing the face’s resistance to being stretched.

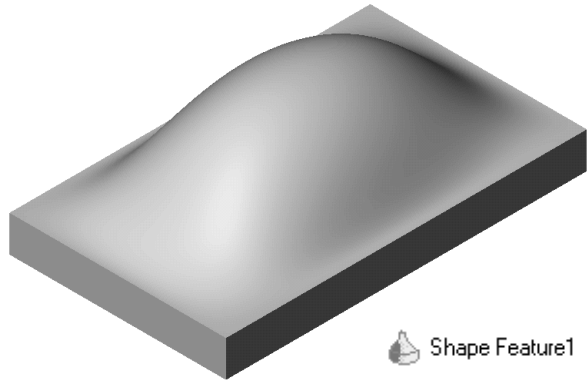


**8 Resistance to bending.**

Change back to the Right view. Set the **Bend** value to **20**, preventing the face from bending easily.

**9 Completed shape.**

Click **OK** to complete the shaping process. A new feature using the shape icon is added to the FeatureManager.

**10 Close the part.**


## Working with Surfaces

SolidWorks 98Plus offers a number of new tools for working with surfaces, modifying the shape of individual faces in a model, and creating advanced shapes that, before this, were not practical.

### Surfaces Behavior

Surface features now behave like other features in a model. You can:

- Select individual edges
- Convert or offset surface edges into a sketch
- Use a planar surface as a sketch plane
- Select the vertices of a surface
- Surfaces, including imported surfaces, can be moved with the **Move/Size**

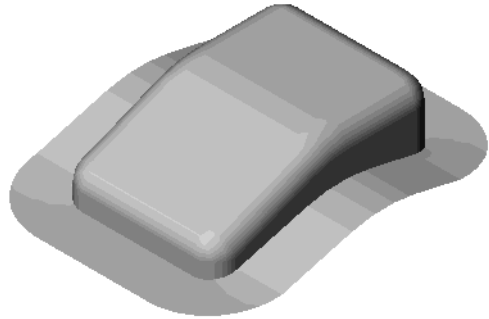
**Features** tool .

### Planar Surface

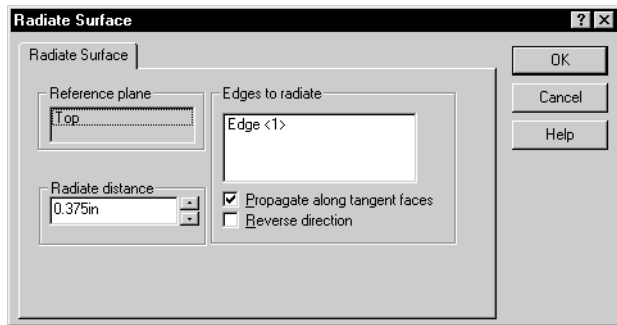
You can create a planar surface from a sketch or model edges. The model edges must be coplanar. Use the command **Insert, Reference Geometry, Planar Surface**.

## Radiate Surface

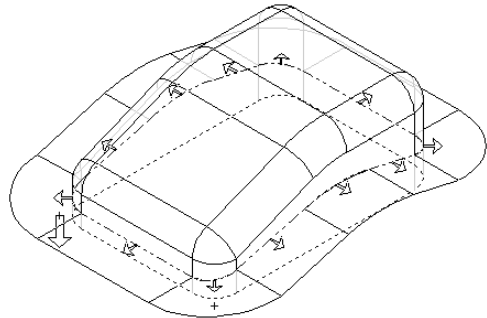
A new option for **Reference Geometry** is **Radiate Surface**. It creates a series of parting surfaces that radiate outward (or inward) from model edges a specified distance. This can be useful in mold making applications.



A **Reference Plane** is chosen to orient the radiate direction. Then, **Edges to Radiate** (in this case created by a **Split Line**) are chosen using propagation. The edges display small arrows indicating the direction the surface will radiate. Lastly, the **Radiate Distance** is set.



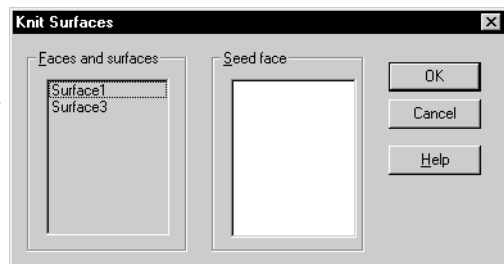
The result is a single surface feature that can be used to cut a mold.



## Knit Surface

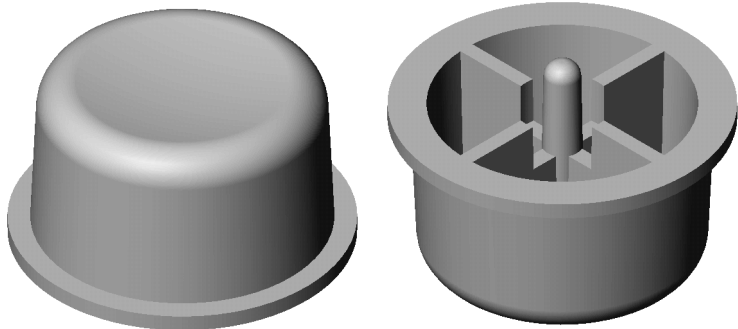
The new knit surface option is used to combine several surfaces into one. The new surface is listed in the FeatureManager design tree as Surface-Knit1.

The dialog is found under **Insert, Reference Geometry, Knit Surface**.



## Example

The following example illustrates how **Radiate Surface** and **Knit Surface** can be used together to create a core for an injection mold. In this case, the part being molded is a small plastic button.

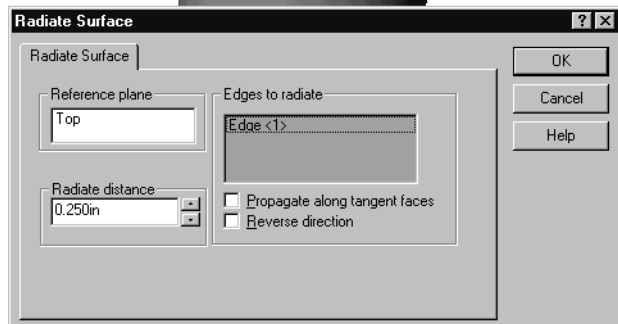
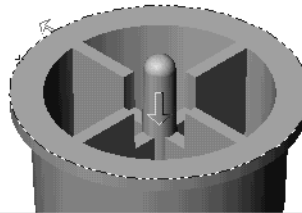


### 1 Open part.

Open the existing part Button in the What ' s New directory.

### 2 Radiate Surface

From the **Insert** menu, choose **Reference Geometry, Radiate Surface**. Select the Top reference plane. Then select the outside edge of the Button as the **Edge to propagate**. Set the **Radiate distance** to **0.25** and click **OK**.

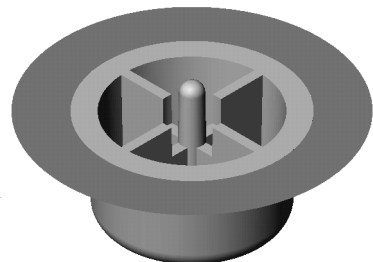


### 3 Results.

The resulting surface is shown here in a different color for clarity.

### 4 Open an assembly.

Open a new assembly and drag the Button into it. For convenience, drop it on the assembly origin so it will be fixed and aligned to the default assembly reference planes.



**Save** the assembly. Name it whatever you want.

### 5 Insert a new part.

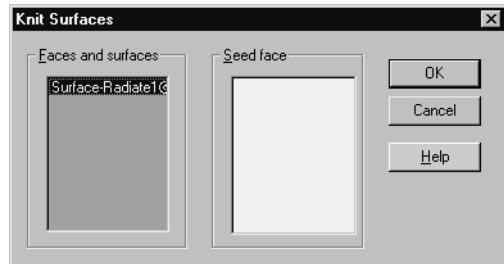
Using **Insert, Component, New**, insert a new part into the assembly. Drop it on Plane1 (Front) of the assembly. When prompted, name this part Core Pin.

**6 Exit the sketch.**

When you insert a new part into an assembly, you are automatically active in a sketch. The next step in the procedure does not require a sketch, so exit it. You should be in **Edit Part** mode, editing the Core Pin.

**7 Knit Surface.**

Orient your view so you can see the underside of the Button. From the **Insert** menu, choose **Reference Geometry, Knit Surface**.

**8 Picking surfaces.**

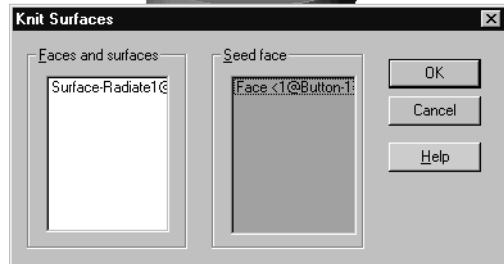
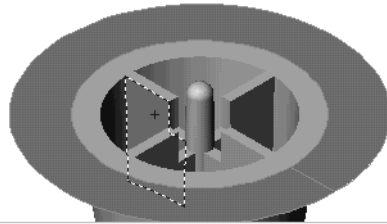
Select the Surface-Radiate feature that is part of the Button. This selection should appear in the **Faces and surfaces** selection list.

This selection should appear in the **Faces and surfaces** selection list.

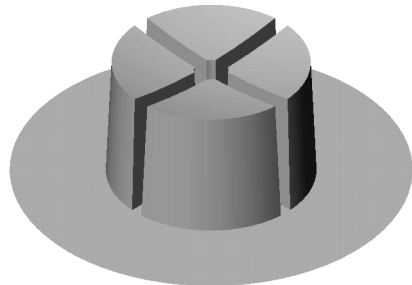
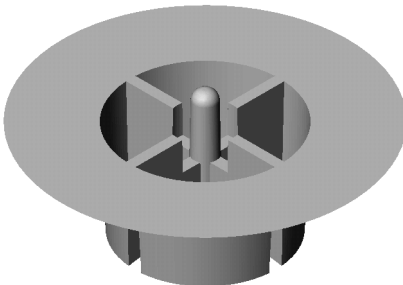
**9 Seed surface.**

Click in the **Seed surface** selection list. Select a face that is part of the inside of the button.

When knitting surfaces, radiated surfaces exhibit a special behavior. When selected in conjunction with a seed surface, they cause the system to select and knit together *all* faces adjacent to it and the seed surface. This eliminates the tedious task of selecting each and every face inside the button.

**10 Results.**

Two views of the results are shown in the illustration below. For clarity, the Button component was hidden prior to capturing the images.



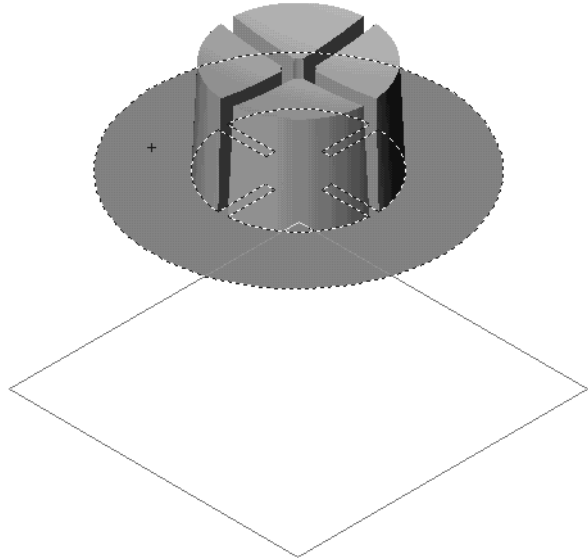


**11 Open the Core Pin part.**

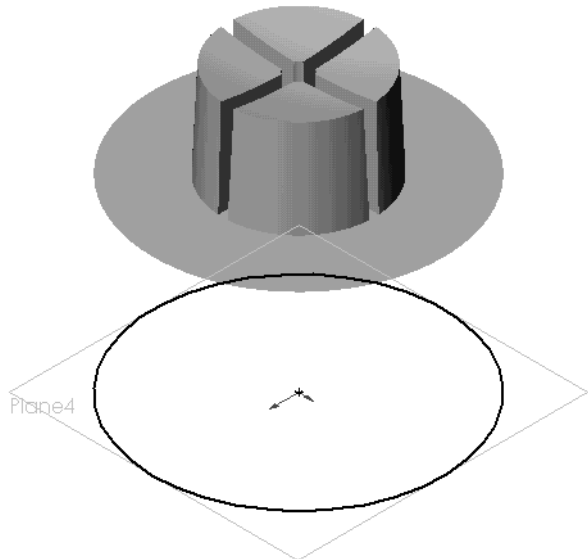
From the right-mouse menu, choose **Open** to open the Core Pin. Granted, the rest of this example could have been done while editing the Core Pin in the context of the assembly, but the illustrations are a little clearer without the Button in the way.

**12 Offset plane.**

Create a plane that is offset from the bottom of the knit surface. Since the face is planar, it can be used to define an offset plane. This is an example of how in SolidWorks 98Plus, surfaces are treated like any other feature.

**13 Sketch.**

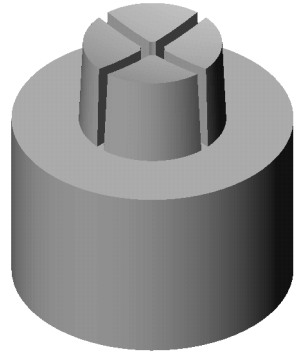
Select the offset plane and open a sketch. Using **Convert Entities**, copy the edge of the knit surface into the active sketch.



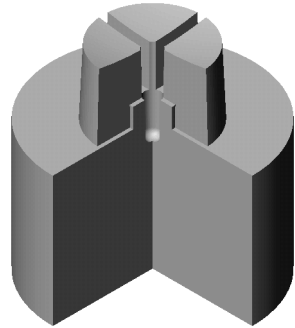
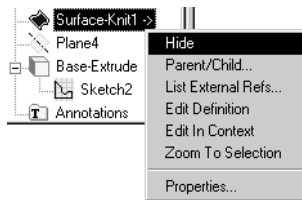
**14 Extrude Up to Surface.**

Extrude the sketch up to the knit surface.

**Important:** When you select the surface in the **Extrude** dialog, pick the knit surface from the FeatureManager design tree. This way the extrusion will go all the way up to *all* faces of the knit surface. If you pick a single face of the knit surface in the graphics window, the extrusion will end at that face.

**15 Results.**

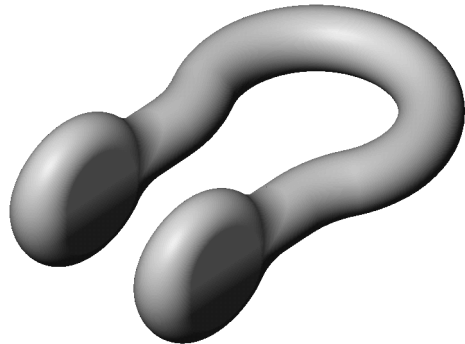
Hide the knit surface to show the finished core pin. The illustration at the right shows a quarter section cut through the pin, revealing the detail.

**16 Close both parts and the assembly.****Another Knit Example**

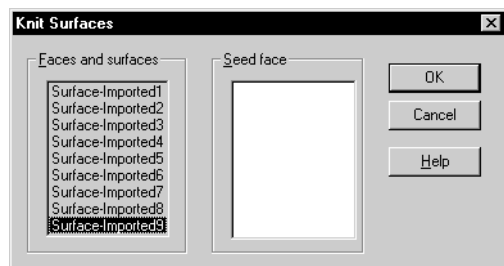
The purpose of this example is to illustrate how to knit several reference surfaces together and then transform them into a solid.

**1 Open part.**

Open the part Knit Surfaces in the What's New directory. The part is a surface model of a forged shackle. It consists of nine imported surfaces.

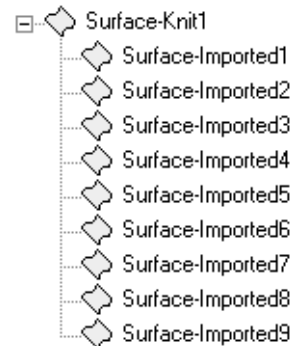
**2 Knit the surfaces.**

Pick **Insert, Reference Geometry, Knit Surface** from the pulldown menu. Use **Shift-select** to pick all nine surfaces in the FeatureManager design tree. You can select them in the graphics window but picking them in the FeatureManager is easier.



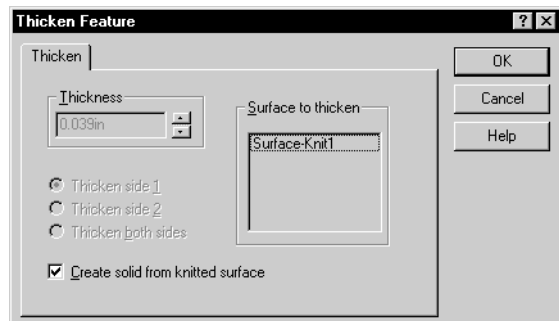
**3 Results.**

The selected surfaces are replaced with a single Surface-Knit feature. If you expand the listing of the feature, you will see the original surfaces embedded beneath it.

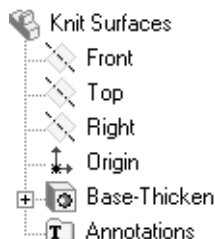
**4 Fill the volume.**

Even though the surfaces are knit into one feature, the volume is still empty. To fill it, select the knit surface and pick **Insert, Base, Thicken** from the pulldown menu.

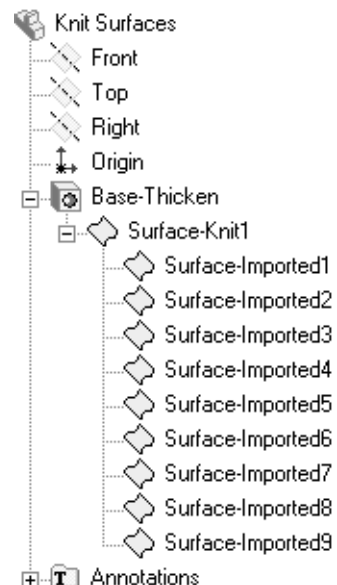
On the **Thicken Feature** dialog, enable the **Create solid from knitted surface** checkbox and click **OK**.

**5 Results**

The part now contains a solid feature named Base-Thicken. Expanding the listing of this reveals its make-up.

**6 Close the part.**

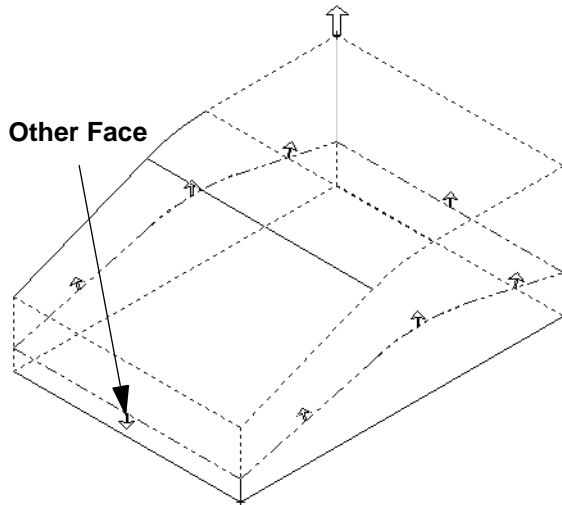
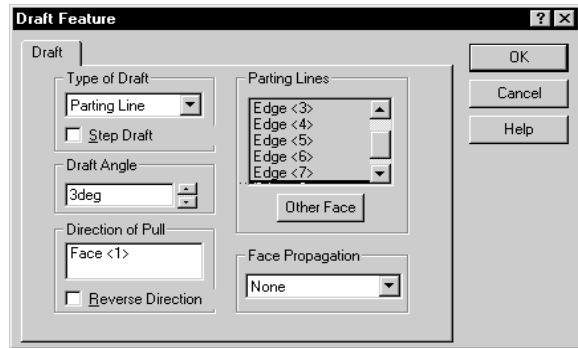
There is no need to save this so close the part without saving.



## Parting Line Draft

The direction of the draft can be changed from the “natural face” for one or more faces. Using the **Other Face** button, you are allowed to reverse the natural direction of the **Parting Line** draft.

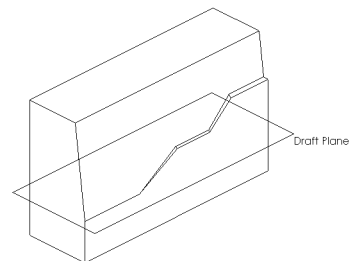
The large arrow indicates the “natural direction”, the direction of pull. When an edge is selected, a small arrow is displayed on that edge, to indicate the face to draft. At this point the **Other Face** option can be selected to get the other face on the common edge. Any number of faces can be reversed in this manner.



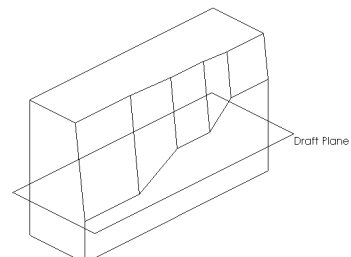
## Step Draft

A new option on the **Parting Line Draft** dialog is **Step Draft** checkbox. When this option is selected, the faces being drafted will rotate with respect to the draft plane *without* breaking up into individual faces at discontinuities in the parting line. When using **Step Draft**, some special conditions apply:

- The pull direction must be defined by a reference plane or planar face. You cannot use a linear edge.
- At least one segment of the split line for each and every face being drafted must lie on the reference plane.
- All segments of the parting line must lie on the same side of the reference plane.



**With Step Draft**



**Without Step Draft**

## Working with Curves

New capabilities for working with curves have been added to SolidWorks 98Plus. These include:

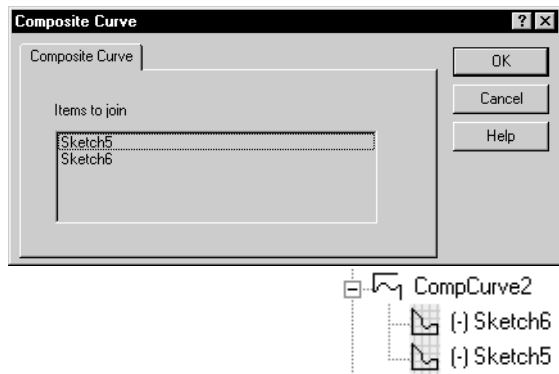
- Using 3D curves as profiles for lofting
- Creating composite curves
- Reference curves in sketches

## Reference Curves

**Reference Curve** geometry such as imported IGES wireframe entities, can now be selected in SolidWorks. It can then be copied into sketches by **Convert Entities** and **Offset Entities**.

## Composite Curves

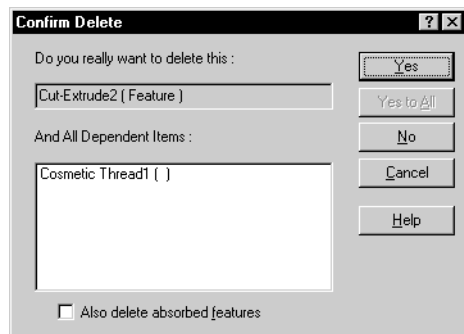
**Composite Curves** can be created using **Insert, Reference Geometry, Composite Curve**. Reference curves, sketch geometry and model edges can be combined into a single curve using this option. This enables you to join multiple sketches into one curve for use as a guide curve when sweeping or lofting.



## Cosmetic Threads

When you add **Cosmetic Threads** to the model, the cosmetic thread is added as a sub-feature of the threaded feature. The thread is dependent on the feature. This represents a change from SolidWorks 98 where the cosmetic thread was embedded in the Annotations folder.

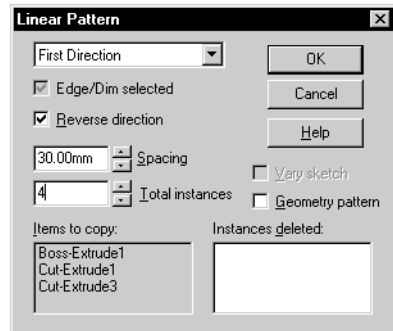
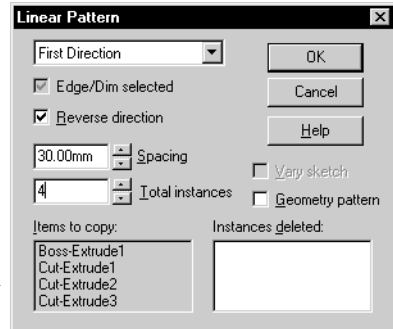
The **Cosmetic Threads** can now be selected graphically for **Edit Definition** or **Delete**. If the parent is deleted, the thread will be as well.



## Patterns

When a pattern consists of more than one feature, you can delete one or more of those features and retain the pattern.

For example, a **Linear Pattern** copies four features. One of the features, Cut-Extrude2, is then deleted from the part. The pattern remains and the deleted feature is removed from the list of **Items to copy**.

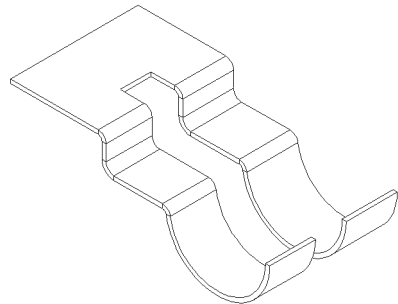
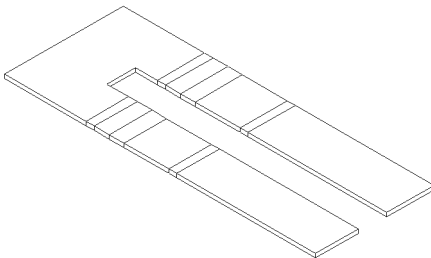


## Sheet Metal

There have been a number of enhancements made to sheet metal functionality including handling of cylinders and large radius rolled parts and form features.

### Unroll Sheet Metal

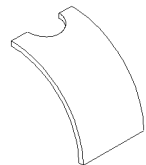
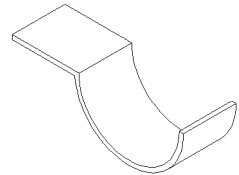
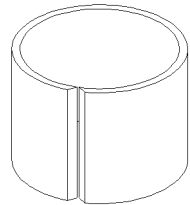
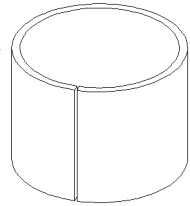
Cylindrical parts and parts with large radius rolled features can now be made as sheet metal parts and flattened.



## Design Guidelines

To model a cylindrical part so that it can be unrolled you must follow these design guidelines:

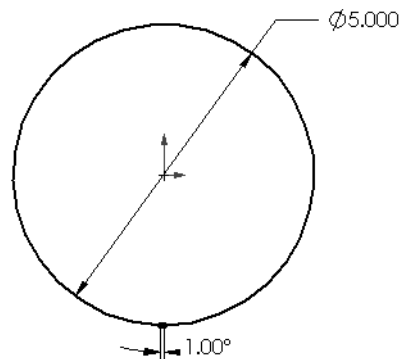
- There must be a slight gap between the ends of the thin material. The best way to achieve this is to sketch and extrude an arc of slightly less than 360°. If needed, you can fill in the gap with a small extrusion later.
- The end faces must be normal to the cylindrical face. That is, they cannot be beveled as shown at the right.
- The cylindrical surface must be tangent to the adjoining surfaces. The part shown at the right will not unroll. However, if you fillet the sketch (as in the illustration on page 3-18) the part unrolls without difficulty.
- The cylindrical face must have four sides. The part at the right has five sides and will not unroll.



## Example

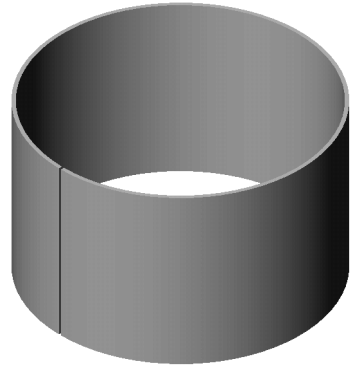
### 1 Sketch the profile.

Use the **Centerpoint Arc** tool to sketch an arc of slightly less than 360°. Dimension the included angle. Use a geometric relation to control the position of one of the ends of the arc. In this case, the right-hand end is positioned a six o'clock or vertically with respect to the circle's center.



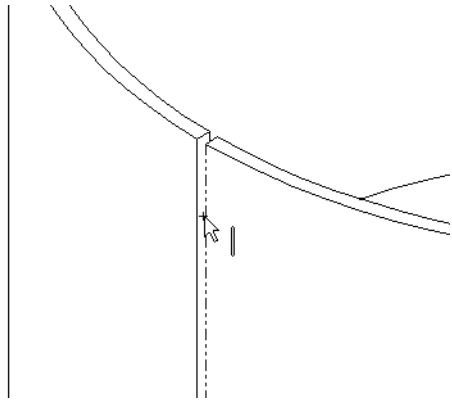
**2 Extrude as a thin feature.**


Set your material thickness to the inside or outside, depending on your design intent. Because of the way the system offsets the material thickness, you do not have to worry about accidentally closing the gap.

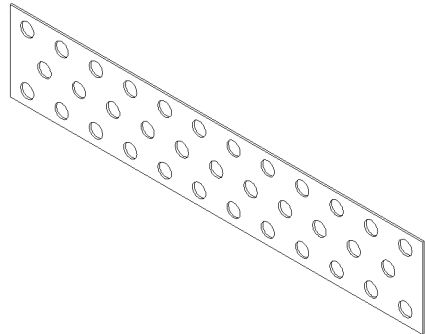
**3 Insert Bends.**

Normally, when you insert bends, you select the planar face that remains stationary when the part is flattened. With a cylindrical part, that is not feasible because the only planar faces are the edges of the thin material. Therefore, you select the linear edge where the gap is in the cylinder. This edge will remain stationary when the cylinder is unrolled.

Because the part is cylindrical, there is no need to enter a default bend radius in the **Insert Bends** dialog.


**4 Flattened the part.**

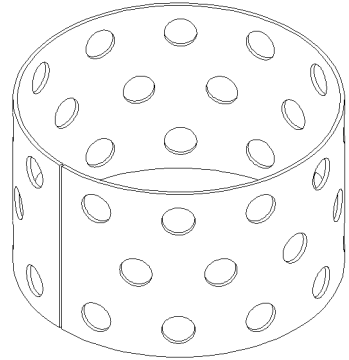
Use the  icon to flatten the part. Create a pattern of holes in the flat part.



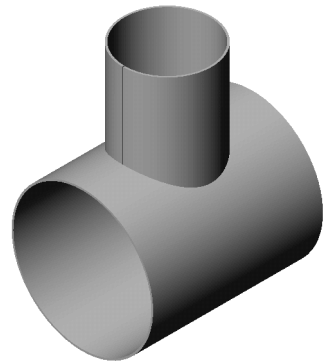


**5 Reroll the part.**

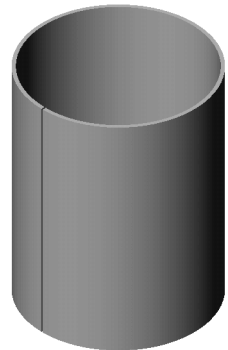
Toggle off the  icon to reroll the part. If you need to make a drawing of the part in both the rolled and flattened states create two configurations. In the configuration representing the flattened part, suppress the `Process-Bends` feature.

**Developments of Intersecting Cylinders**

Unrolling cylinders can also be used to create flat pattern developments such as the case of the smaller of these two intersecting cylinders.

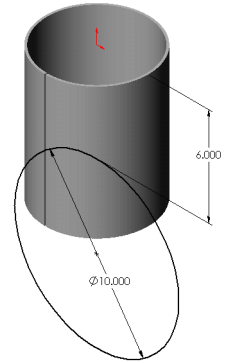
**1 Model the cylinder.**

Following the procedure given in the preceding example, model the smaller cylinder. Remember to leave a gap in the side of the cylinder so you can unroll it.

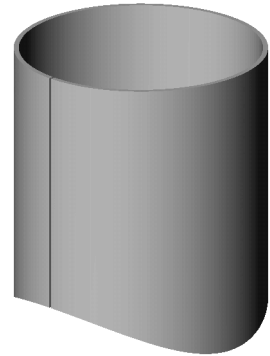


**2 Sketch the second cylinder.**

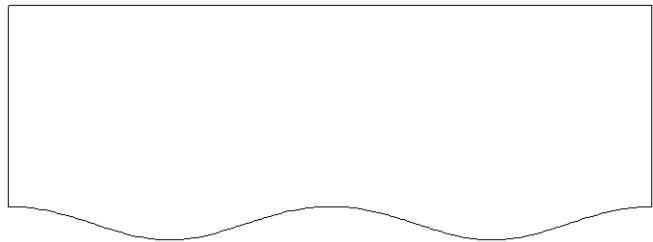
Sketch a circle representing the size and location of the second, larger cylinder.

**3 Cut.**

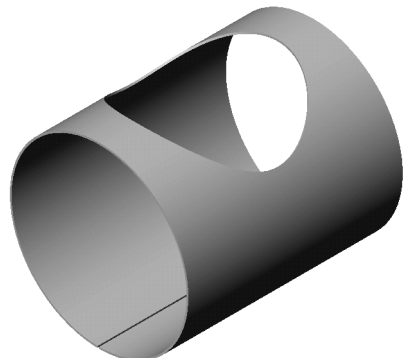
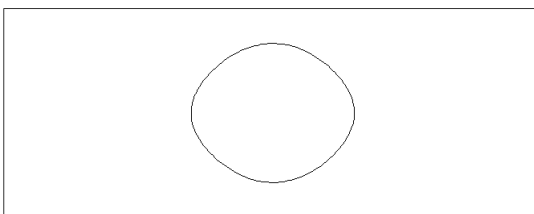
Extrude a cut to sculpt the cylinder to the shape of what would be the larger cylinder.

**4 Unroll.**

As described previously, unroll the cylinder to create a developed flat pattern.

**The Other Cylinder**


If you cut a hole in the other cylinder, SolidWorks 98Plus can unroll that, too.

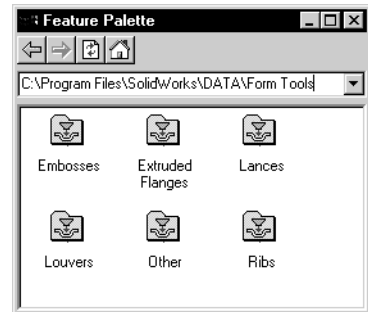
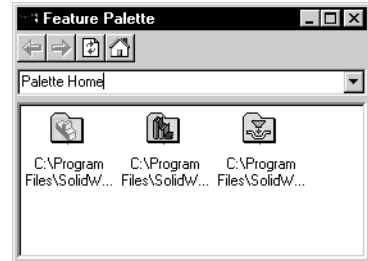


## Form features

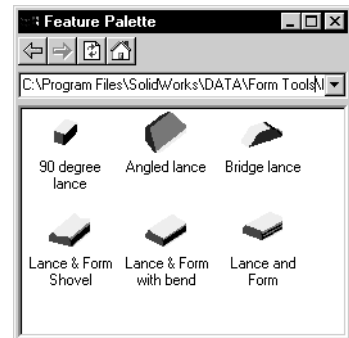
With SolidWorks98Plus, you can add **Form features** such as louvers, lances and flanges to sheet metal parts. A library of **Forming Tools** is available through the Feature Palette window. You can also create your own form features.

### Library of Forming Tools

By setting the path for **Palette Forming Tools** in the **Tools, Options, External References** dialog, you can access forming tool parts and directories. The forming tool directories and parts are marked with a folder and  symbol.



The individual forming tools are parts specially designed for use in creating form features. They are intended to be used like other palette features: to be dragged and dropped onto a face of a model. In this case the model must be a sheet metal part.

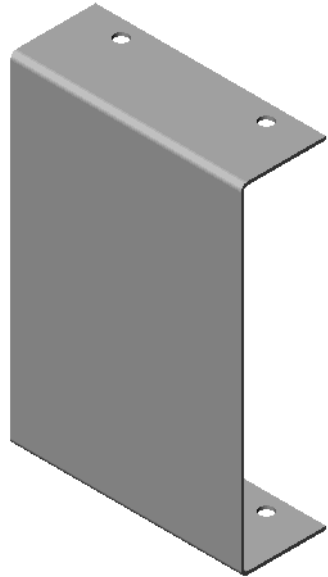


**1 Open the part.**

Open the existing part form features in the What's New directory. The part is a sheet metal part with holes in the flanges.

**Add a Louver**

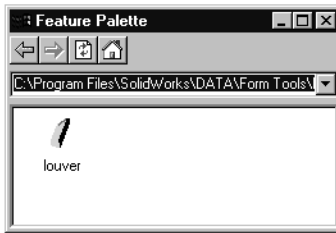
The Louver forming tool creates a form feature that pushes the material out and creates a hole.

**2 Drag and drop.**

Change the view orientation to *inside\_iso*. We will add the form feature to the *inside* face of the model.

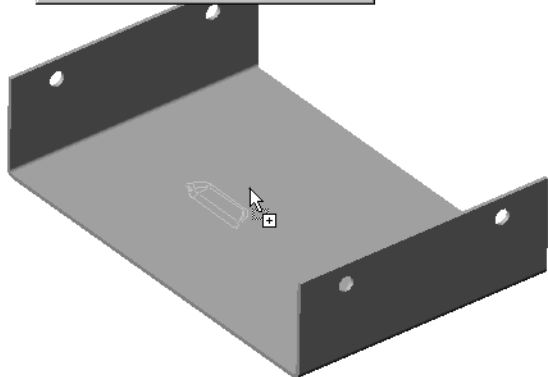
**3 Louver.**

Open the Feature Palette window and click to the **Forming Tools** folder named **Louvers**. Drag and drop the forming tool *louver* onto the inner face of the model.



**Note:** The form features can be pushed through from the drop face or pulled out from it. By default, SolidWorks will push them into the drop face. If you press the **Tab** key before you drop the feature, it will be pulled out. Pressing **Tab** key repeatedly will toggle this behavior with the preview graphics updating accordingly.

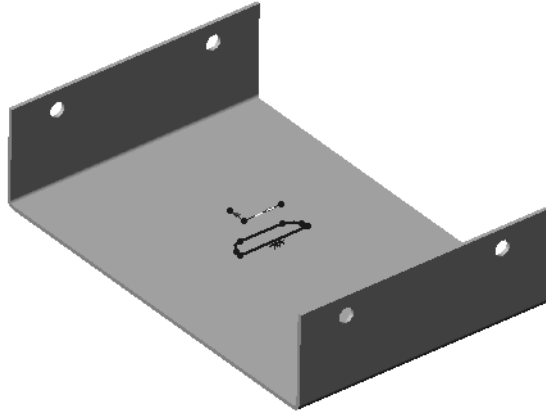
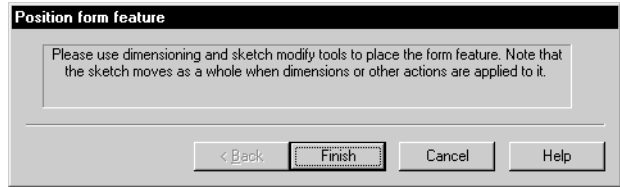
Since we want to push the form feature *out* from the drop face, there is no need to press the **Tab** key.



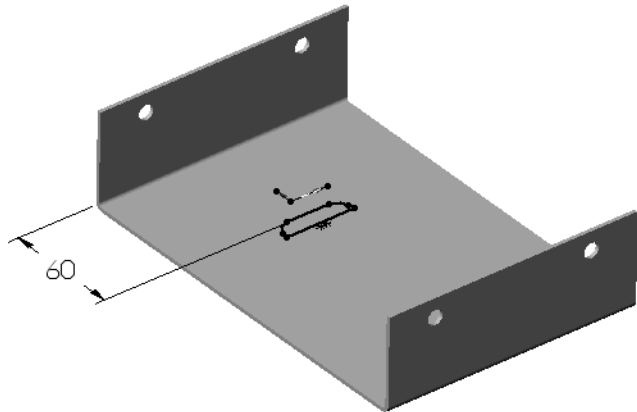
**4 Orient the sketch.**

Position the forming tool by adding dimensions or relations to the sketch.

Using **Modify Sketch**, rotate the sketch **-90°** to orient it properly.

**5 Fully Define the sketch.**

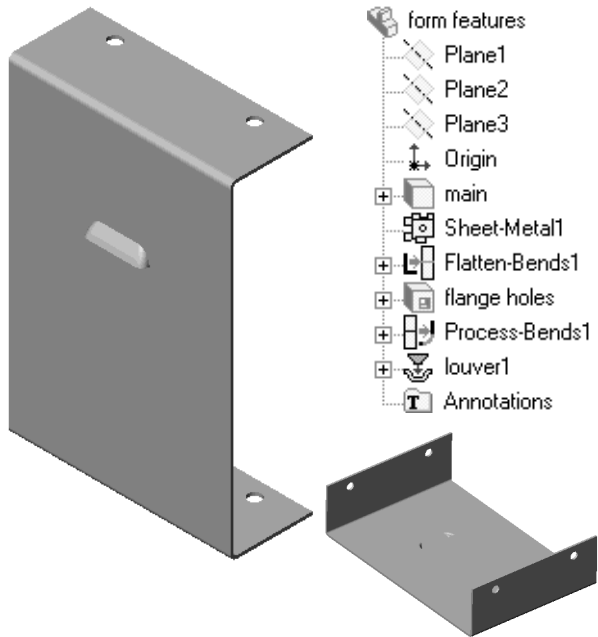
Add a **Collinear** relation between the short centerline and the reference plane Plane3. Add a dimension as shown to fully define the sketch.



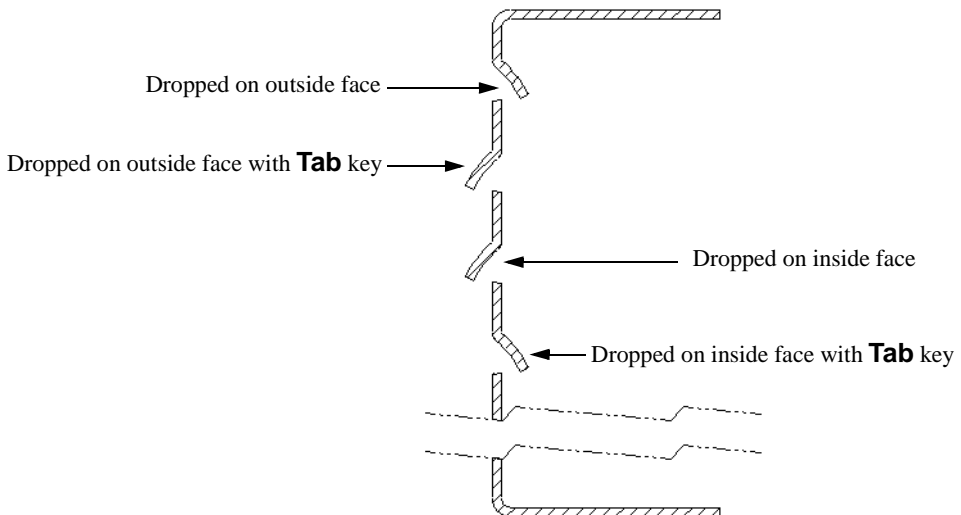
**6 Click Finish.**

Click the **Finish** button to add the form feature to the model. The form feature appears in the feature manager with the name `louver1`.

The form feature can also be added in the flat state and will appear there. Simply roll back the model to before `Process-Bends1` and add the feature.

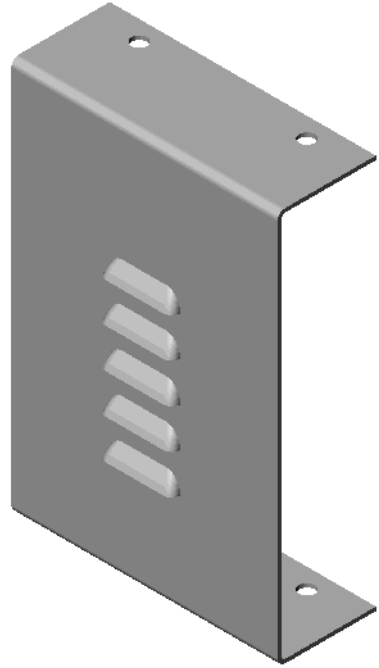
**Direction of the Forming Tool**

When adding a feature like a louver, you have the option of dropping it on the inside or outside face. By default, the tool punches through the face on which it is dropped. However, by using the **Tab** key, you can reverse the default behavior. It determines whether the louver is punched in or out.



**7 Linear Pattern.**

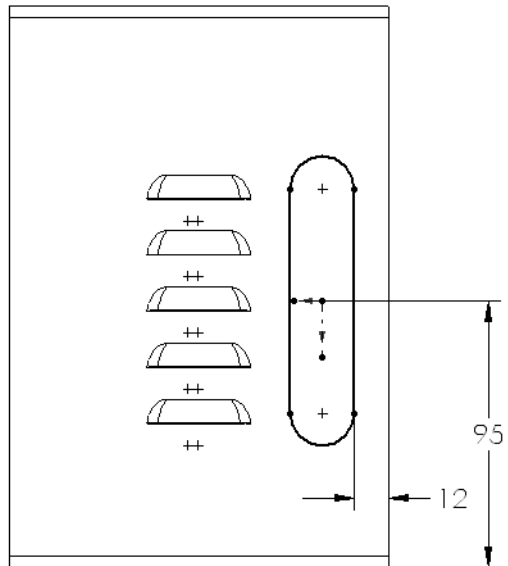
Pattern the form feature to create a total of **5** louvers spaced **20mm** apart.

**Add a Rib**

The Rib forming tool is used to create a rounded shape that pushes the material out but does not cut it.

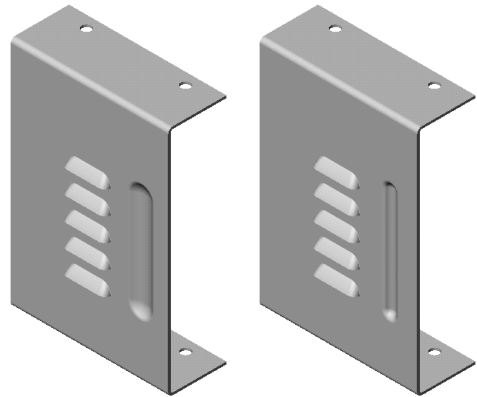
**8 Drag and drop the Single Rib.**

Drag and drop the Single Rib forming tool from the Ribs folder onto the *front* face of the model. Add the dimensions shown.

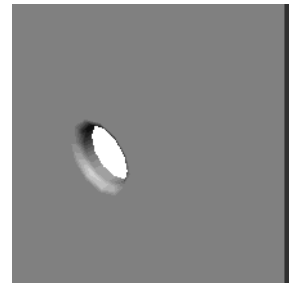


**9 Finished rib.**

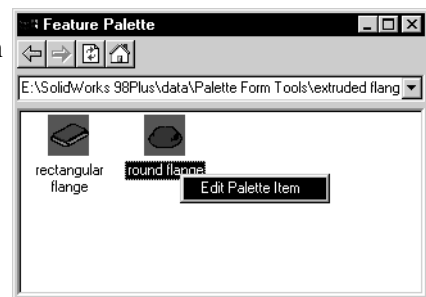
Click the **Finish** button to add the feature. Change the size of the feature by double clicking on it and setting the **20mm** rib width dimension to **10mm**. Rebuild the model.

**10 File the part.****Creating Your Own Forming Tools**

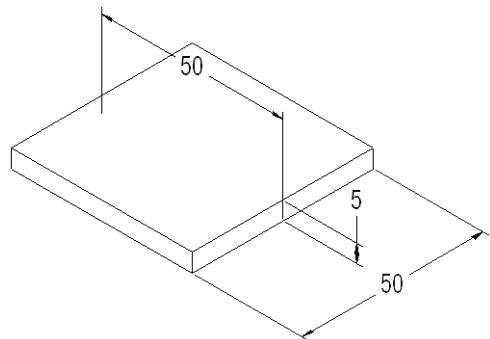
You can create your own forming tools by following the methods used in creating the standard ones. Let's look closely at a typical form feature part, the round flange, feature by feature.

**1 Open the library part.**

Position the cursor over the round flange icon in the Feature Palette window and pick **Edit Palette Item** from the right-mouse menu. (It is the only choice.) This will open the corresponding library part in your SolidWorks window.

**2 Rollback to Base-Extrude.**

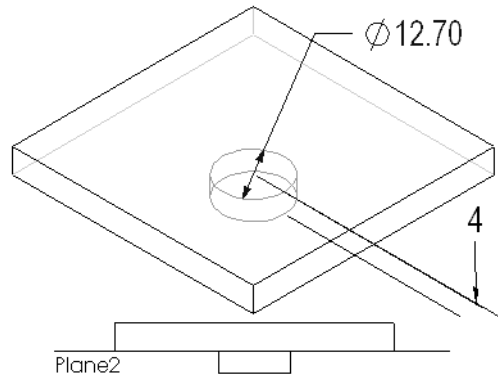
The base feature of the form represents the sheet metal it will be attached to.



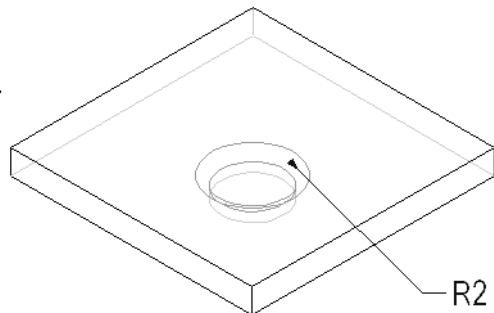


**3 Boss-Revolve1.**

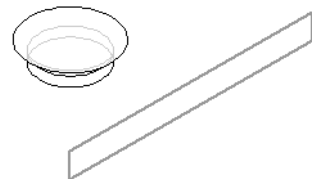
All of the form feature geometry must fall below Plane2 (Top) in the model. The cylinder was created as a revolved feature of **360°**.

**4 Fillet3.**

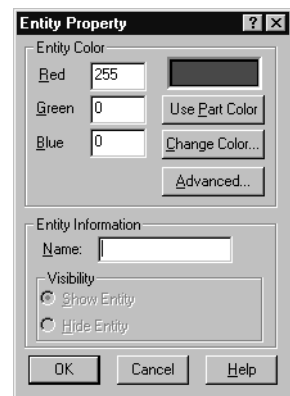
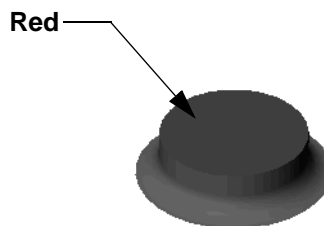
The fillet was added to the revolved feature where it meets the base feature.

**5 Cut-Extrude2.**

The entire base feature was removed using a cut. The existing edges of the original Base-Extrude were used to form the sketch for the cut.

**6 Face color.**

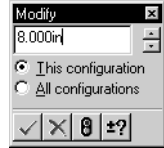
The system determines which face punctures the metal by its color. Since the circular face opposite the fillet is intended to puncture the sheet metal, it is given the color **Red** (255,0,0). This is done using **Face Properties, Change Color**. This is required. It has to be red.

**7 Close the part.**

## Configurations

In previous versions of SolidWorks there was a dichotomy between the behavior of configurations created directly and those created by design tables: only design table configurations could control parameter values. In SolidWorks 98Plus, you can directly create different configurations that reflect different parameter values.

When you double click on a dimension of a part that has configurations, a new dialog appears. The two options on the **Modify** dialog specify whether the change is to affect just **This configuration** or **All configurations** in the part.

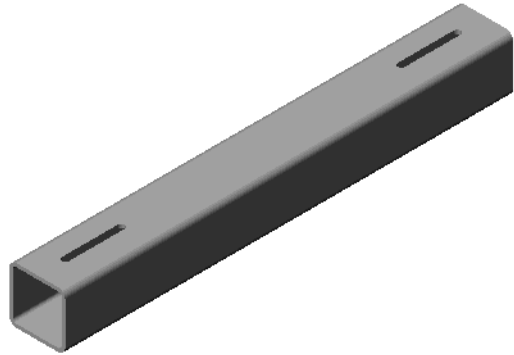


### 1 Open the part.

Open the existing part `config_values` in the What's New directory.

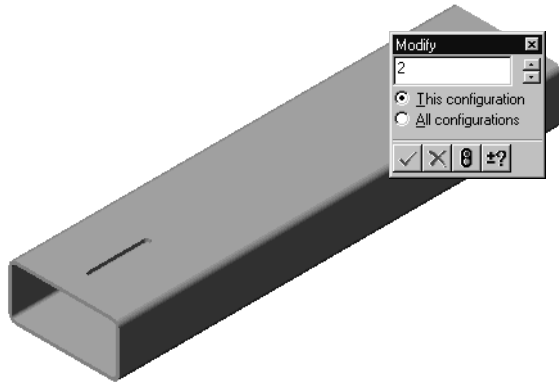
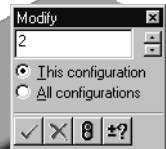
### 2 Change configurations.

Switch from the Default configuration to the one named **Altered Form**.



### 3 Value change.

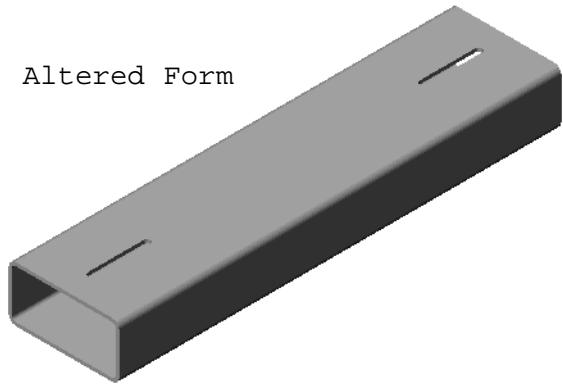
Double-click on the base feature. Change the horizontal **1** inch dimension to **2** inches for **This configuration** only. Rebuild the model.



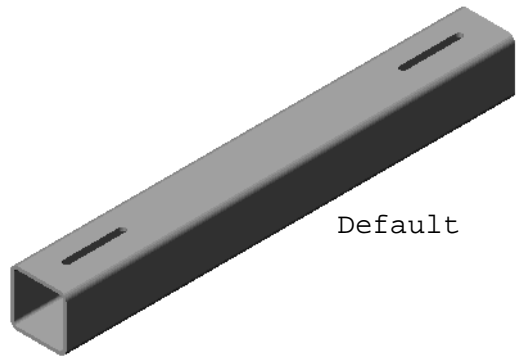
#### 4 Results.

The current configuration is the only one that displays the dimension change. The configuration Default has no change to that dimension.

Altered Form



Default



### Library Features

Library Features now support several new options:

- Limit access to dimensions by editing dimension access
- Simplification of the library feature
- Support of **Sweep** and **Loft** features in the library feature

### Edit Dimension Access

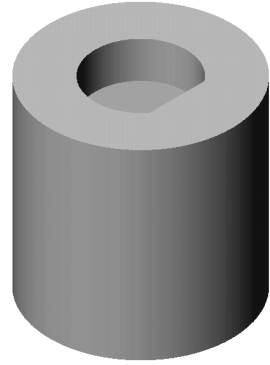
The option **Edit Dimension Access** is provided to limit what dimensions, if any, appear in the library feature after it is inserted into a part. The restrictions exist *only* if the library part is inserted via the Feature Palette window. By regulating which dimensions appear, changes to the library feature can be restricted.

This **Edit Dimension Access** dialog is available for *all* parts, not just library parts.

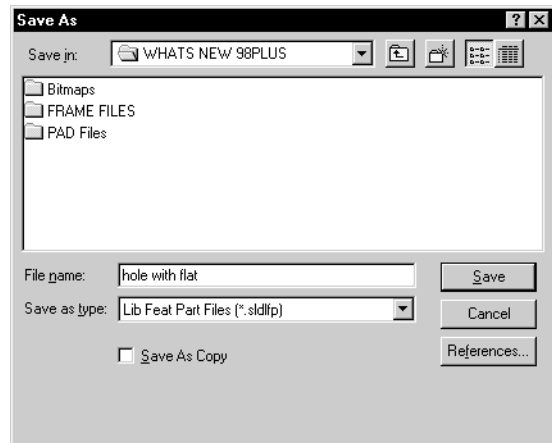
In this example, you will create a new library feature and determine which dimensions appear when it is used.

**1 Open the part.**

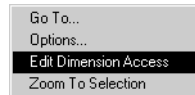
Open the existing part dim access in the What ' s New directory. The part is a cylinder with a single cut feature. — a D-shaped hole.

**2 Save library feature.**

Select the cut feature (Cut - Extrude1) and use **File, Save As** to create a library feature named hole with flat.sldlfp and click **Save**.

**3 Edit Dimension Access.**

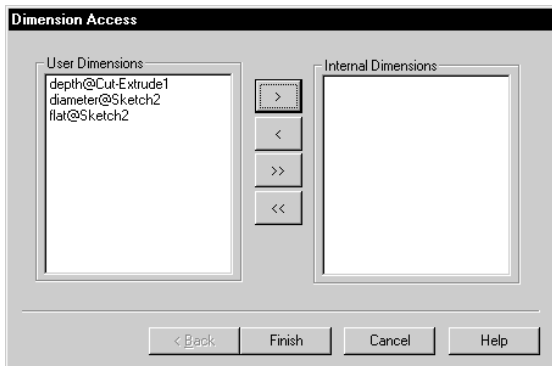
From the top level feature (hole with flat) choose **Edit Dimension Access** from the right-mouse menu.

**4 Dimension Access dialog.**

The **dimension access** dialog contains two columns, **User Dimensions** and **Internal Dimensions**. All the dimensions in the library feature will appear in one column or the other, but not both.

**User Dimensions** are those that appear in the part when the inserted library feature is double-clicked.

**Internal Dimensions** *do not* appear in the part when the inserted library feature is double-clicked.



## 5 Controls.

The center buttons control the movement of the dimensions between columns. One or more dimensions (**Ctrl**-select and **Shift**-select can be used) can be moved to the other column. The buttons work as follows:



**Move selected dimension(s) into Internal Dimensions column**



**Move selected dimension(s) into User Dimensions column**





**Move all dimension(s) into Internal Dimensions column**

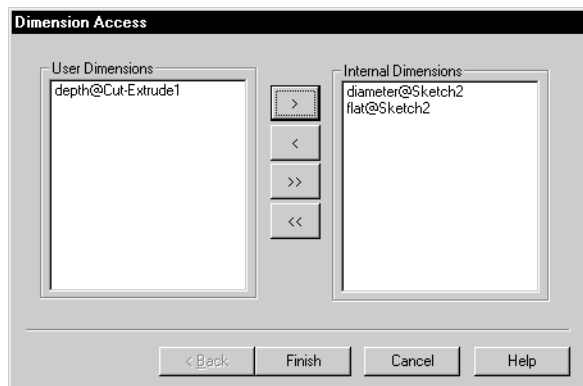


**Move all dimension(s) into User Dimensions column**

**Tip:** Library features will many times have a large number of dimensions, only a few of which you want users to be able to change. It is often times more efficient to move *all* the dimensions into the **Internal Dimension** column. Then move only the ones you want users to edit back into the **User Dimension** column.

## 6 Settings.

Move all the dimensions to the **Internal Dimensions** column using the  button. Move the `depth@Cut-Extrude1` dimension back to the **User Dimensions** column using the  button. Click **Finish** to retain the settings.

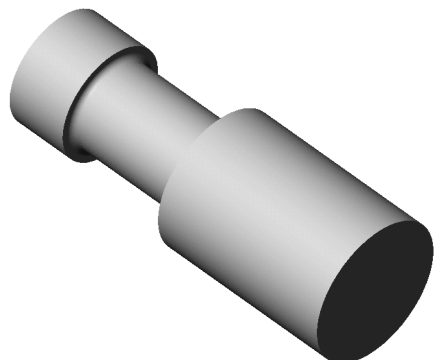


## 7 Close the library feature.

Close the library feature, saving the changes.

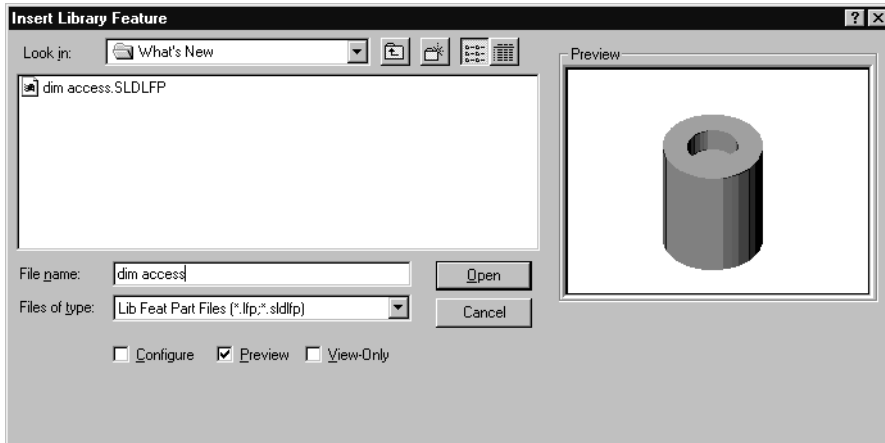
## 8 Open another part.

Open the existing part `insert libfeat` in the `What's New` directory. The part is a simple turned shaft.



## 9 Insert Library Feature.

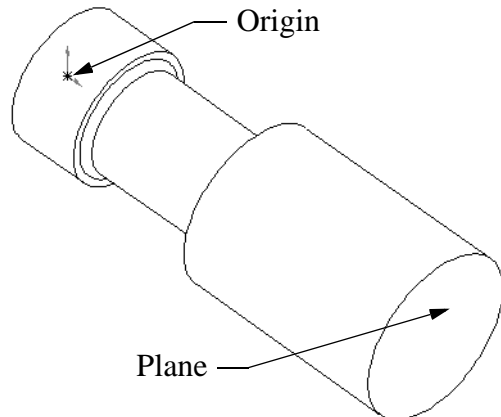
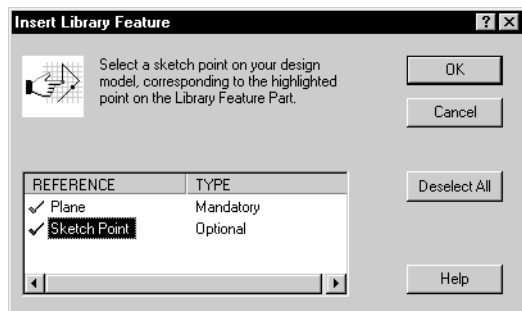
Select the library feature hole with flat.sldlfp. Click **Open**.



## 10 Reference selections.

For the **Mandatory Reference Plane** select the circular face of the model. For the **Optional Reference Sketch Point** select the **Origin** of the part from the FeatureManager design tree or graphics window.

Click **OK** to add the library feature.



**11 Dimensions.**

Double-click on the feature LibFeat1 and only the **User Dimensions** will appear.

**Important**

All the dimensions can still be accessed by expanding the features listed under the library feature.

**12 Undo the insert.**

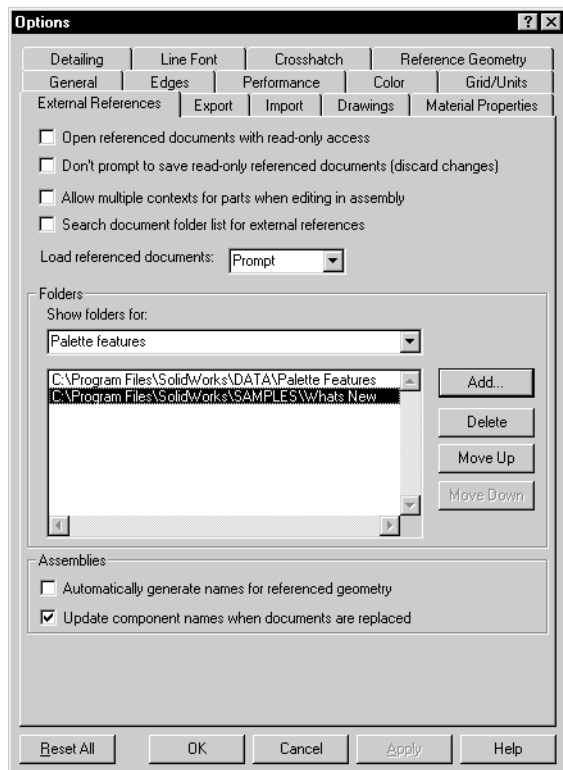
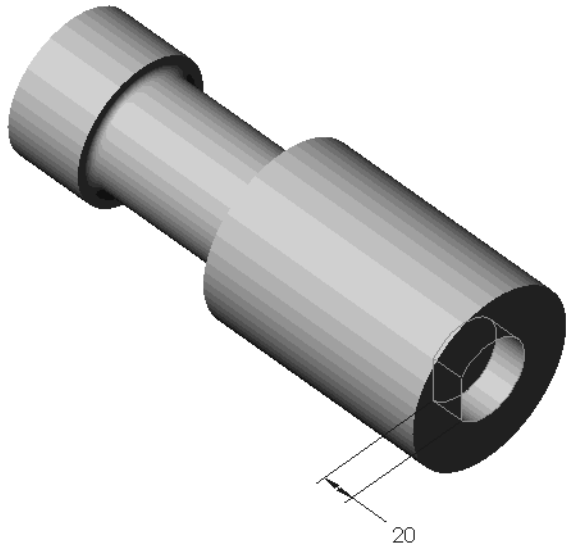
**Undo** the insertion of the library feature. We will now insert the same feature using the Feature Palette window.

**13 Palette features.**

Using the **Options** dialog, set the **External References** for **Palette features** to include the What's New directory.

Click **OK**.

This will allow the feature palette to see the library feature hole with flat as a palette feature.

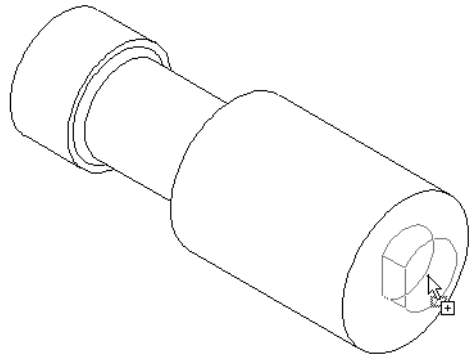


**14 Feature Palette.**

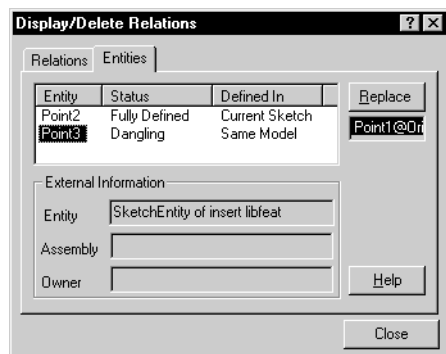
Open the Feature Palette window using the **Tools** menu. Double-click the **What's ' New** folder. The hole with flat should appear.

**15 Drag and drop.**

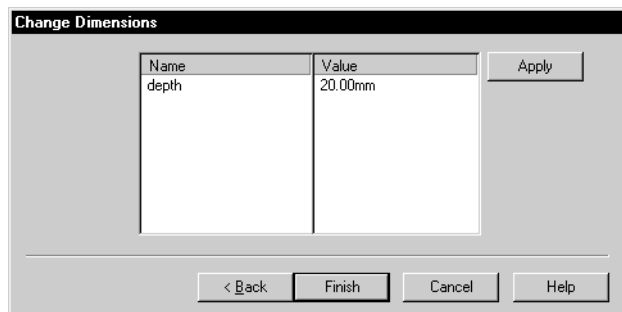
Drag and drop the palette feature onto the circular end face.

**16 Repair dangling relation.**

**Replace** the dangling centerpoint relation with the Origin using **Display/Delete Relations**. Click **Close** on this dialog and **Continue** on the **Edit the Sketch** dialog.

**17 User Dimension.**

The sole dimension that appears in the **Change Dimensions** dialog is the one tagged as the **User Dimension**. The others are inaccessible. Click **Finish**.



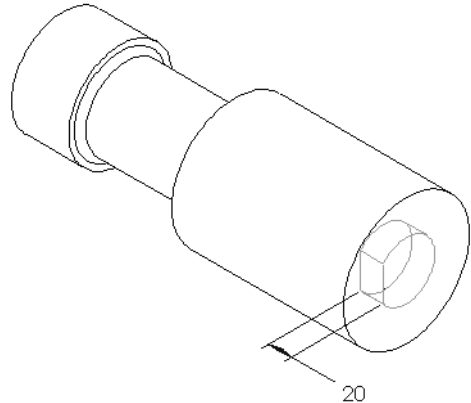


**18 Double-click the feature.**

Double-clicking on the copied palette feature shows that only the **User Dimension** is accessible.

**19 Close the part.****Conclusion**

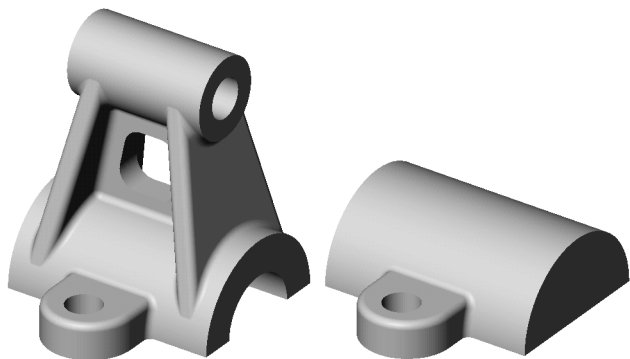
As you can see from this example, you have better control over the user's access to a library feature's dimensions if it is inserted using the Feature Palette window rather than **Insert Library Feature**.

**Simplification of Library Features**

If you are working on a part and you have just finished making a feature that you would like to save as a library feature, you can do so. Select the feature(s) that you want to be included in the library feature and use **File, Save As** to make the library feature. When you do this, the system will prompt you asking if you want to simplify the library feature. If you click **Yes**, the system will remove all features except the base feature and the features you selected. Keep in mind that this might cause some rebuild errors in the library feature if it was built with references to features the system deleted during simplification.



Consider the part shown at the right. The last features built were the mounting tab on the base and its fillet. To save this as a library feature, select the boss and the fillet. The system will strip out all the other features in the part, leaving just the base feature, the extruded boss, and the fillet.



## Design Tables

The syntax rules governing **Design Tables** have been changed to support **Assembly Design Tables**. Refer to *Module 5: Assembly Enhancements*.

Although the old syntax is still supported, this new method is preferred.

- New column headers are supported. These include STATE and COMMENT. There are others but they apply only to assemblies, so they won't be listed here.
- These new column headers are preceded by the '\$' symbol and are sometimes followed by an '@' symbol and feature name.

The format of the design table is the same as in previous versions of SolidWorks. Cells A2, A3, A4, and so on contain the configuration names. Cells B1, C1, D1, and so on contain the dimension name, STATE or COMMENT headers. The cell A1 remains blank and appropriate values propagate the others.

	A	B	C	D	E
1		BodyLength@Sketch1	\$STATE@Fillet5	\$STATE@Fillet4	\$COMMENT
2	pconfig1	2	suppressed	unsuppressed	
3	pconfig2	3	s	suppressed	no fillets
4	pconfig3	4	suppressed	unsuppressed	
5					
6	\$USER_NOTES	overall length			

The following is a description of the new column headers:

### ■ Dimension

The name of a dimension can be used as a column header. Use the **FullName** of the dimension as shown in the dimension's **Properties**. The dimension uses standard naming like BodyLength@Sketch1. This is the same as in previous versions of SolidWorks and has not changed for SolidWorks 98Plus. However, new in Solidworks 98Plus is that the use of dimension names in design tables is no longer case sensitive.

### ■ State

The state of a feature is whether or not it is suppressed. The syntax for the STATE column header is: \$STATE@Feature\_name.

The cells in this column can have values of suppressed (abbreviated s), or unsuppressed (abbreviated u).

### ■ Comments

Using the COMMENT header, you can enter text into a column and it will be ignored by SolidWorks. The syntax is for the column header is \$COMMENT.

### ■ User Notes

This can be used as either a column or row header. The syntax is \$USER\_NOTES.

1 Open the part.

Open the existing part part\_dt in the What ' s New directory. The part has been swept and shelled.

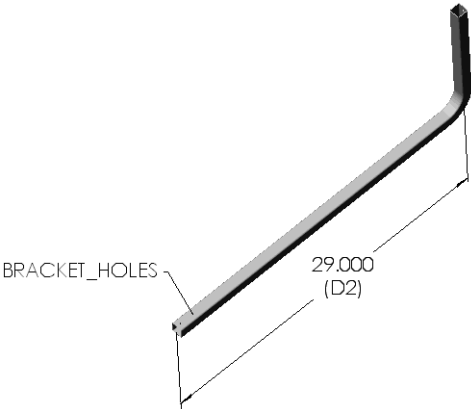


2 Design Table.

Create a Excel spreadsheet, naming it part\_dt.xls. Create two configurations in the table, HOLES and NO\_HOLES. The dimension D2@Sketch1 is taken from the base feature. BRACKET\_HOLES is a cut feature.

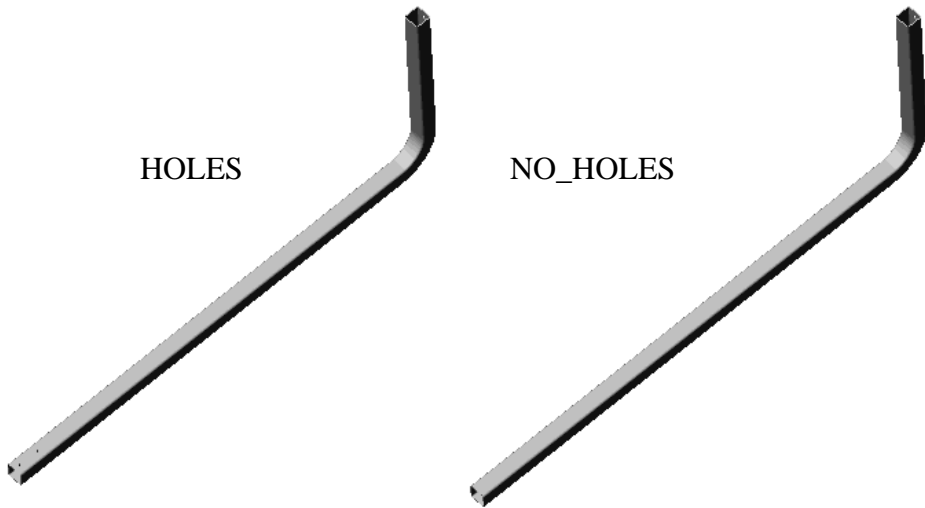
	A	B	C
1		D2@Sketch1	\$STATE@BRACKET_HOLES
2	HOLES	29	UNSUPPRESSED
3	NO_HOLES	31	S

Insert the design table into the part.



**3 Configurations.**

Two new configurations are added by the design table.

**4 Close the file.**

## **Module 4**

# **Drawing Enhancements**



## Importing Model Dimensions

Now you can import model dimensions into all views of the drawing at once. The dialog option **Import Dimensions Into All Views** must be checked.

SolidWorks adds dimensions to section and detail views first, then to standard views.

## Detail and Section Views

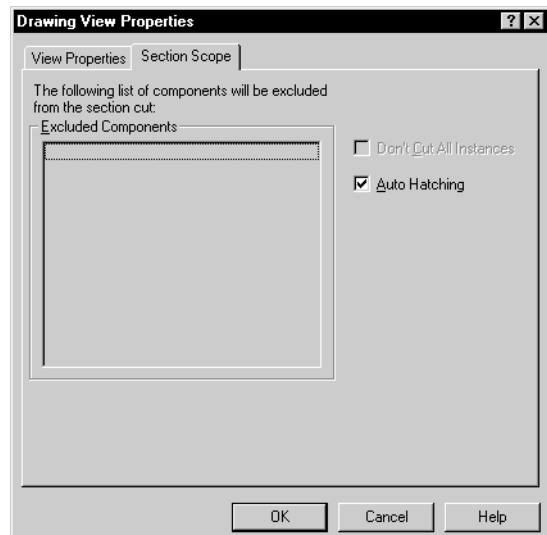
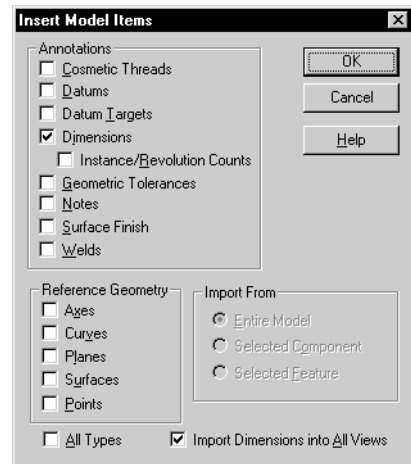
Now **Detail** and **Section** views can be cut and pasted to another sheet within a drawing. Use **Edit, Cut** and **Edit, Paste**, the cut and paste icons, **CTRL+X** and **CTRL+V**, or drag and drop the view from one FeatureManager window to the other.

You can also copy and paste section and detail views to other *drawings*. However, these will only paste if their parent view is on the new drawing as well. For example, assume that on Draw1 you have a Front view and a section view of it. If you copy the Front view and paste it onto Draw2, you can then copy the section view to Draw2 as well. In fact, you can copy both views at the same time if you want to.

## Auto Hatching of Section Views

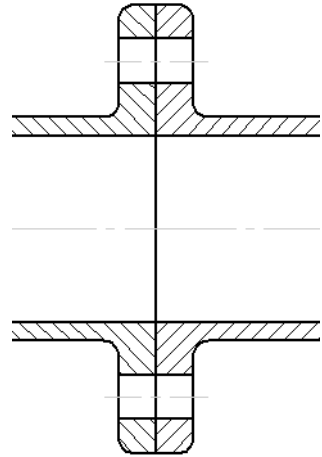
In SolidWorks 98Plus, assemblies that are sectioned can have **Auto Hatching** to vary the cross hatching between adjacent components in the assembly.

When the assembly is sectioned, the **Section Scope** dialog appears with a new option, **Auto Hatching**, checked by default.



The hatching appears in the section view using different angles on adjacent components.

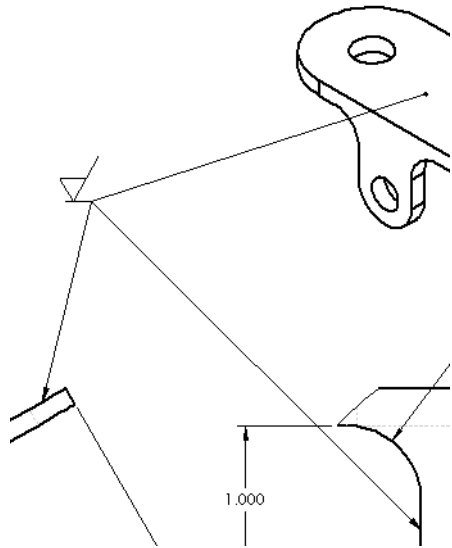
The resulting cross hatching can be changed. In fact, you can use **Ctrl-select** to select and change *all* the cross hatching of a single part at once.



## Multiple Leaders

Multiple leaders can now be created and deleted for **Surface Finish**, **Weld**, **Balloon** and **Geometric Tolerance** symbols in addition to **Notes**.

They behave like multiple attachment **Notes**. **Ctrl-drag** an endpoint and drop it on an edge or face to add more leaders. Click the leader's arrow and the **Delete** key to remove it.



## Dynamic View Activation

As mentioned in *Module 2: General Information*, you can set an option on **Tools, Options, Drawings** to enable automatic drawing view activation. This eliminates the need to double-click in a view before adding view-related items such as section lines or detail circles. However, there may be times when you want to temporarily disable this function. To accomplish this, you have the option of locking and unlocking the sheet or view focus. Please note: if you have dynamic view activation *disabled*, locking and unlocking sheet/view focus is not available and does not apply.



## Lock Sheet Focus

When dynamic view activation is enabled, one view on the drawing is always active. For those times when you do not want any view active, for example when adding notes on the drawing sheet, you can lock the focus on the sheet. Do this by picking **Lock Sheet Focus** from the right-mouse menu. The status bar of the drawing window indicates the current focus **Editing Sheet (Locked)**.

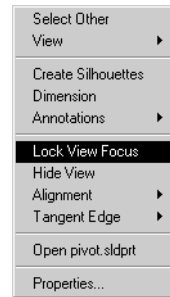


To return to dynamic view activation you can:

- Double-click on any view
- Pick **Unlock Sheet Focus** from the right-mouse menu

## Lock View Focus

**Lock View Focus** is for those times when you want a single view to be active, and stay active, even though the cursor might move around the drawing. To set the focus to a particular view, position the cursor inside the view and pick **Lock View Focus** from the right-mouse menu. The status bar of the drawing window indicates the current focus **Editing Drawing View (Locked)**.



To return to dynamic view activation you can:

- Double-click on any *other* view
- Pick **Unlock View Focus** from the right-mouse menu

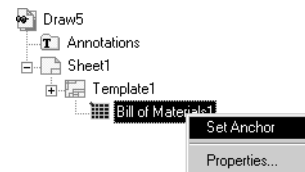
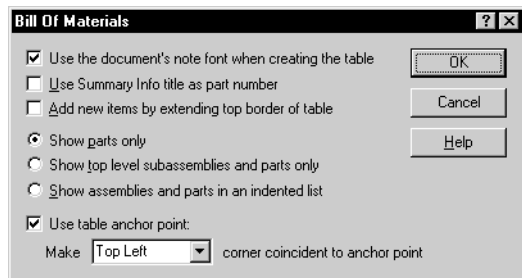
## Bill Of Materials Enhancements

With SolidWorks 98Plus, you can anchor the bill of materials block to the drawing sheet.

When you insert the bill of materials, **Use table anchor point** gives you four choices: **Top Left**, **Top Right**, **Bottom Left** and **Bottom Right**.

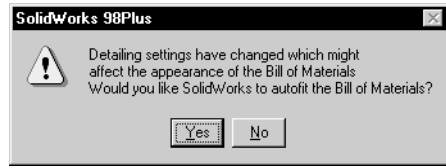
These determine which corner of the bill of materials snaps to the anchor point on the template.

To define the anchor point on a template, expand the Template feature in the drawing's FeatureManager window. There you will see a Bill of Materials feature. Pick **Set Anchor** from the right-mouse menu. The system will switch into **Edit Template** mode and you can pick a location on the drawing template. This



location will become the anchor point used by the bill of materials when it is inserted. This anchor point can be saved in the template using **Save Template**.

If the BOM is set to **Use the document's note font when creating the table**, changes to the **Note Font** setting will bring about this message. Click **Yes** to have SolidWorks resize the BOM to the new text size.



You can manually add items to the bill of materials. This enables you to list items that do not require components in the assembly. Examples of such items include oil, paint, safety wires, and thread sealant. This also enables you to include standard fasteners without having to include every nut, bolt, and washer in the assembly.

To add items to the BOM, double-click on the bill of materials. This will start Excel. Select the *row* containing the keyword \$END. Using the Excel command **Insert, Rows**, add as many new rows as you will need. You can then type the information you want into the cells.

	A	B	C	D	E
1	ITEM NO.	QTY.	PART NO.	DESCRIPTION	\$\$END
2	1	1	bracket		
3	2	1	Yoke_male		
4	3	1	spider		
5	4	1	U-joint_pin1		
6	5	3	U-joint_pin2		
7	6	1	Yoke_female		
8	7	1	crank-shaft		
9	8	1	crank-arm		
10	9	1	crank-knob		
11			\$\$END		

## Drag Multiple Views

Now multiple drawing views can be dragged simultaneously on the drawing sheet. On the screen, use **Ctrl**-select to identify them. In the FeatureManager design tree, use either **Ctrl**- or **Shift**- select. Drag one of the views by the visible edge and they will all move.

## Broken View Enhancements

**Broken views** have been enhanced so that they can show:

- Sections lines and detail geometry
- Temporary axes
- Cosmetic threads

## Cosmetic Thread Enhancements

**Cosmetic Threads** have new behavior in drawings.

- In section views hidden cosmetic threads will no longer be displayed
- Dimensions of cosmetic threads can be imported into drawing views
- Thread display is copied with the view

## GOST Standard

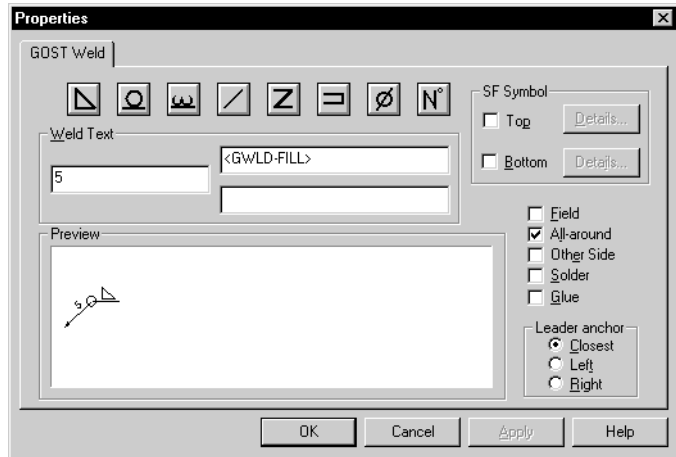
The GOST Russian dimensioning standard is now available from **Tools, Options, Detailing, Dimensioning Standard**. Some of the details are shown below.

### Choosing the GOST Standard

Choose the **GOST** standard from the **Options, Detailing, Dimensioning Standard** option. It is listed as one of the choices in the dropdown list.

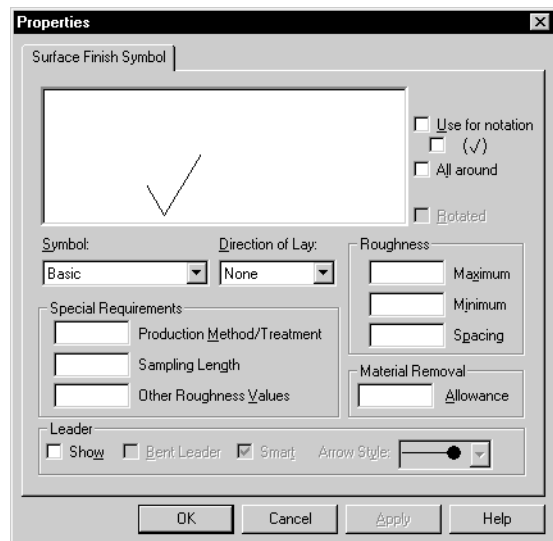
### GOST Weld Symbol

The **GOST Weld Symbol** dialog includes the weld symbols as buttons across the top of the dialog.



### GOST Surface Finish Symbol

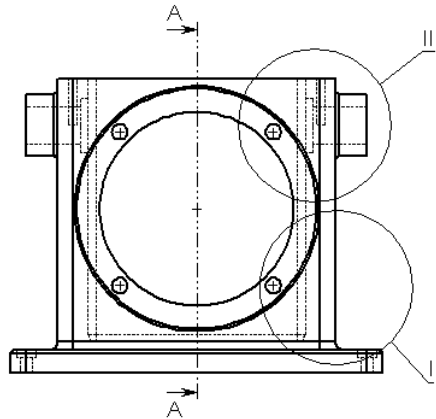
The **Surface Finish Symbol** dialog includes the option **Use for notation**.



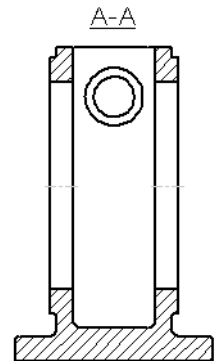
## Section and Detail Views

The **GOST** standard for section and detail views is shown here. In the parent view, the **Section Line** A-A and **Detail Views** I and II are shown.

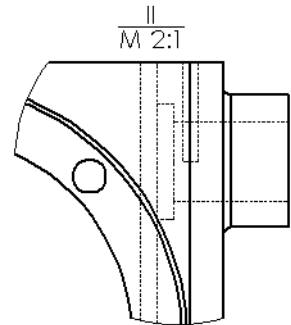
The **Section** arrows are offset from the ends of the section line by **3mm**. The **Detail** circles include leaders to the text and an underlined roman numeral as an identifier. The text can be dragged.



The **Section View** shows the name, A-A, underlined and above the section view.

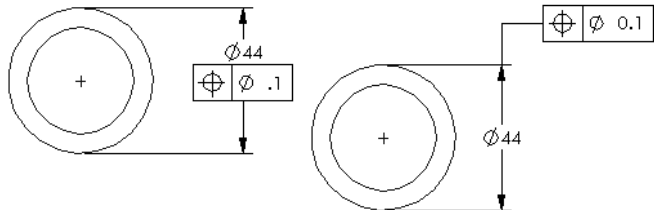


The **Detail View** displays the section name underlined with the scale listed below.



## Geometric Tolerance Enhancements

**Geometric Tolerance** frames attached to dimensions can now be moved outside the witness lines. A leader extends to the frame through the dimension.



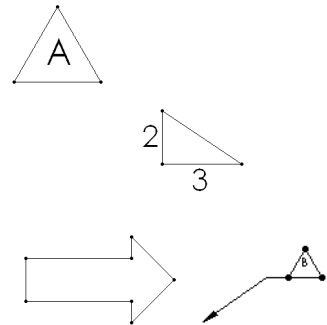
## Suppressed Dimensions

If a feature is suppressed in a part, the dimensions of that feature are suppressed in any drawings of it.

## User Defined Symbols

You can now create your own drawing symbols and save them as a library for general use. These symbols are strictly 2D, created using sketch entities, notes or a combination of both. The file type is \*.sldsym, and it is created in its own special drawing. You can access symbols within the library to change them.

The custom symbol can have a leader or optionally a bent leader attached at the symbol origin. These options are added after the fact using the **Properties** of the symbol.



### 1 Open a drawing.

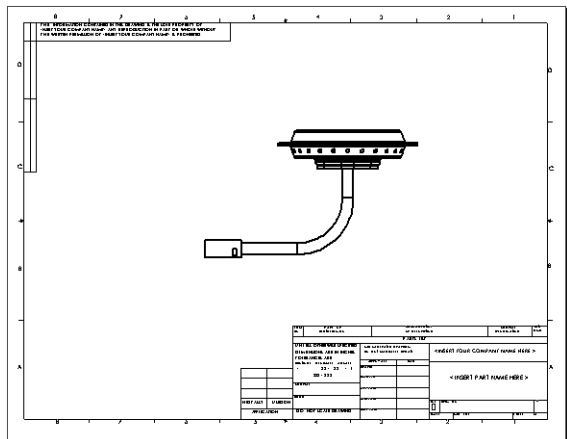
Open an existing drawing named custom symbols.

## Creating the Symbol

The symbol is created in a separate drawing, a special type for symbols. The drawing will be created when the **New** symbol is initiated.

### 2 New symbol drawing.

Use the option **Tools, Custom Symbol, New** to start the symbol editor. A new drawing appears. The new drawing has no template.



### 3 Insert a note.

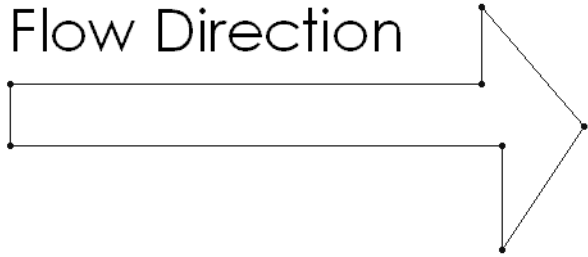
If the symbol will contain both text and sketch geometry, you may want to gauge the sketch to the size of the text.

Flow Direction

In this example create text **Century Gothic, Regular**, sized at **14 points**. The text string should read Flow Direction.

**4 Sketch an arrow.**

Using lines, sketch an arrow as shown.

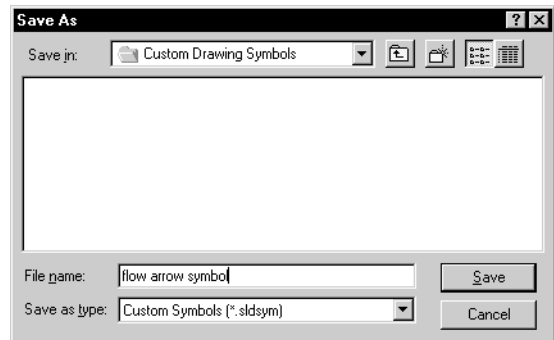


Use **Equal** and **Collinear** relations to shape the arrow properly.

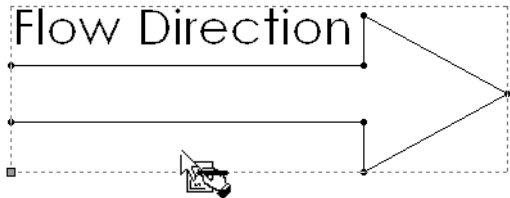
Note that dimensions and crosshatching will *not* appear in the finished symbol but construction geometry will.

**5 Save the symbol.**

Click the **Save** option and name the symbol flow arrow symbol. The extension \*.sldsym is added automatically. Click **Save**.

**6 Saved symbol.**

The symbol has now been saved to the library. It is a single piece of geometry with an attachment point at the left lower corner of the enclosing box.



**Tip:** Zoom in tightly on the symbol before you save the file. It will give you a better looking bitmap image for the preview graphics when you go to insert the symbol.

**7 Close the drawing.**

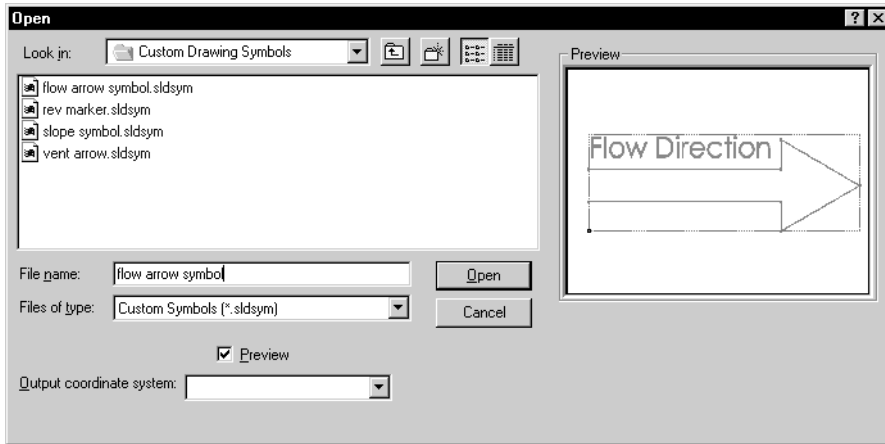
Close the symbol drawing, saving it.

## Using the Symbol

Once saved, the symbol can be used in any drawing sheet. It is inserted by name and moved as a single object.

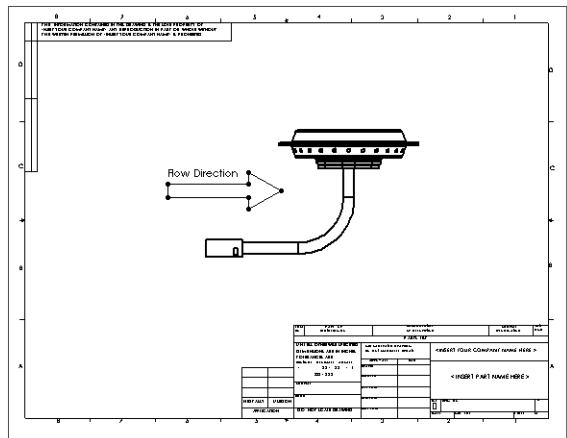
### 8 Insert Custom Symbol.

Click on the drawing in the approximate location where you want the symbol to appear. Click **Insert, Annotations, Custom Symbol** and select the file flow arrow symbol from the browser. Click **Open**.

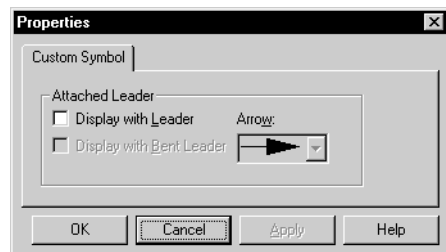


### 9 Symbol.

The custom symbol is placed on the drawing sheet where you clicked prior to inserting it.

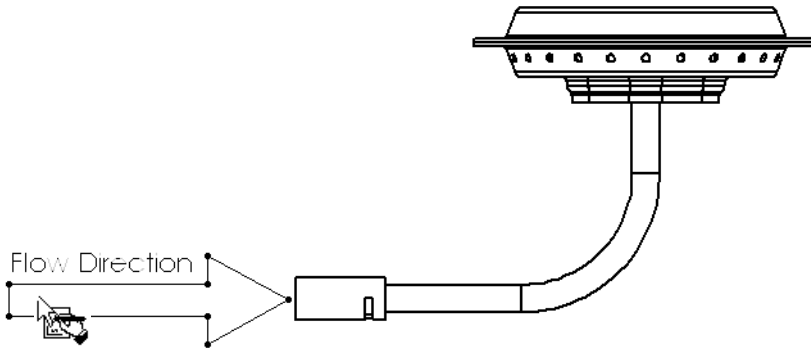


The symbol appears without a leader. If you want to add a leader, use the **Properties** of the symbol.



## 10 Place the symbol.

Drag and drop the symbol near the venturi tube inlet. The writing hand symbol appears when the cursor is over custom symbol.

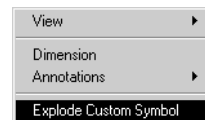


## Editing a Custom Symbol

If the symbol contains text you can edit it directly on the drawing. Simply double-click on the note and retype the characters in the highlighted text string that appears. This will retain the symbol but update the text.

If you want to change the symbol's graphics, it must be edited using **Tools, Custom Symbol, Edit**. The **Edit** option brings up the **Open** dialog. Browse for the symbol you want to edit and click **Open**. The system will bring you back into the drawing of the custom symbol.

Since the symbol is a single object, it must be broken down into its constituent pieces to be changed. Do that using the **Explode Custom Symbol** option on the right-mouse menu.



## 11 Copy and paste.

Select the custom symbol, copy it and paste it onto the drawing sheet. The changes will be made to the copy.

## 12 Changes.

Edit the note text in the copied symbol. Double-click the Flow Direction text and type Air Vent. Click outside the symbol to finish editing.



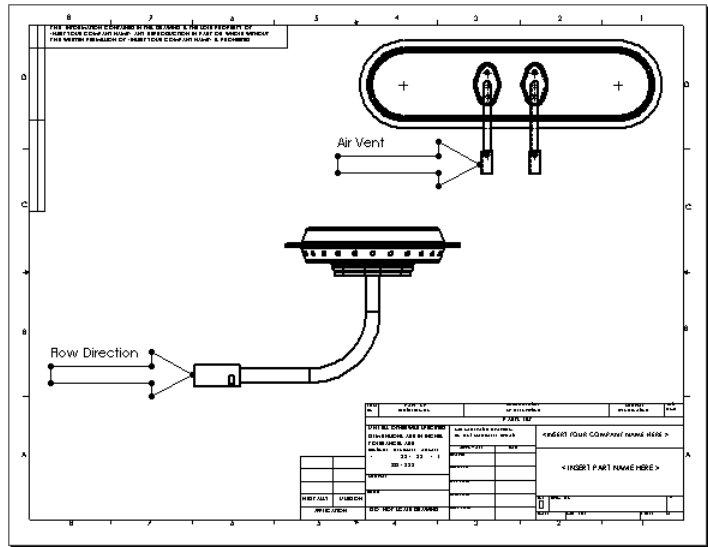


**13 Open the referenced file.**

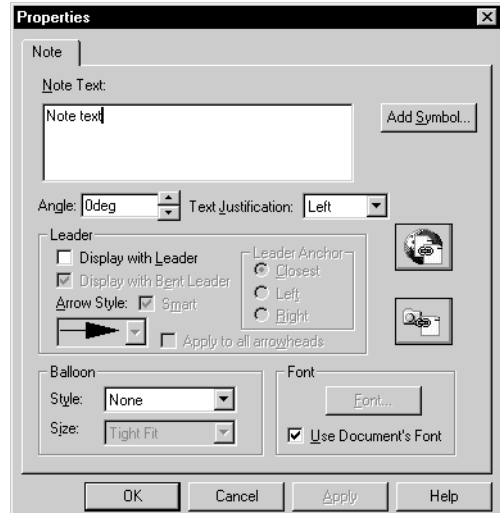
Open the assembly by picking **Open burner subassy.sldasm** from the right-mouse menu.

**14 Add a view.**

Add a view to the drawing sheet using the named view **vents** and a **Scale** of **1:4**. Position it in the upper right corner of the sheet. Move the copied symbol to the view.

**15 Close the drawing.****Justified Text Notes**

A **Text Justification** option has been added to the **Note Properties** dialog. The text can be justified to the **Left**, **Right** or **Center** positions.



## Document Properties

SolidWorks 98Plus has expanded the basic system **Properties** available in a drawing.

### Properties

The **Properties** can be used with notes and the **Link to Property** option. They are described below.

- **SW-File Name**

The SolidWorks part/assembly/drawing name (without the file extension) is output into the note. See **External model reference** below.

- **SW-Template Size**

Outputs the sheet or template size (A, C, A2, etc.).

- **SW-Sheet Scale**

The sheet scale of the attached view is output.

- **SW-Current Sheet**

Listed as an integer value (1, 2, 3...) in order.

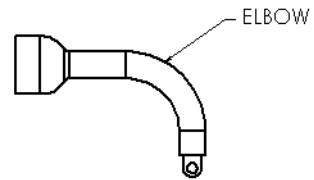
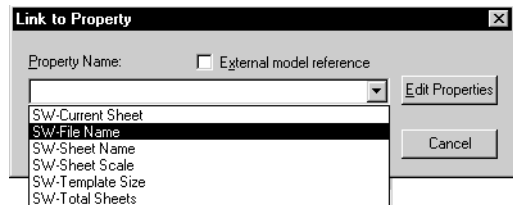
- **SW-Total Sheets**

An integer indicating the total number of sheets in the drawing file.

- **SW-Sheet Name**

The name of the sheet as you create it.

Using these properties, you can create drawing notes such as “**Sheet 1 of 5**” where the “**1**” and “**5**” are read from the drawing file itself. The note will update if the number of sheets changes.



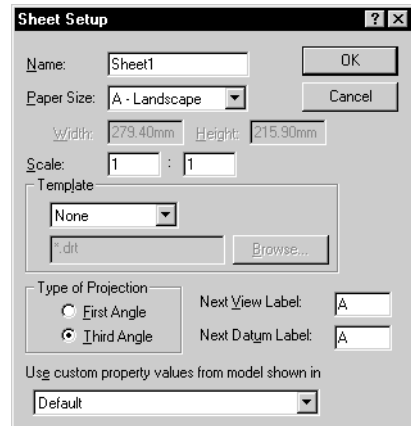
The **External model reference** checkbox has a different effect depending on whether a note is attached to the view or a drawing. A drawing can contain views of different parts and/or assemblies. If the note is attached to the drawing sheet, **External model reference** causes it to use the first reference in the drawing sheet. That is, the part or assembly referenced by the first view as listed in the drawing's FeatureManager tree.

If the note is attached to the drawing sheet and **External model reference** is *not* checked, it uses the drawing file name.

If the note is attached to a view, it uses the name of the file referenced by that view.

## Multiple References

For situations when more than one part/assembly is displayed on one sheet, the **Sheet Setup** dialog provides some options. The pulldown beneath **Use custom property values from model shown in** has the values **Default**, **Drawing View 1**, **Drawing View 2** and so on. This allows you to choose the source of the properties used.



### 1 Open a new drawing.

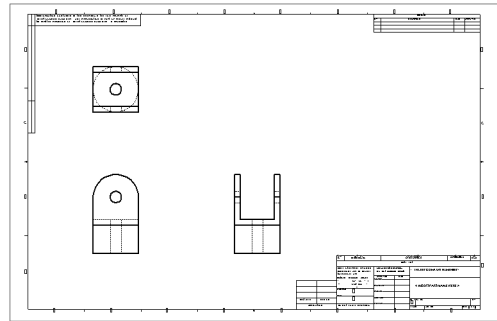
Open a new drawing, B-Landscape, using the standard template.

### 2 Open a part.

Open the existing part **Pivot** located in the What 's New directory.

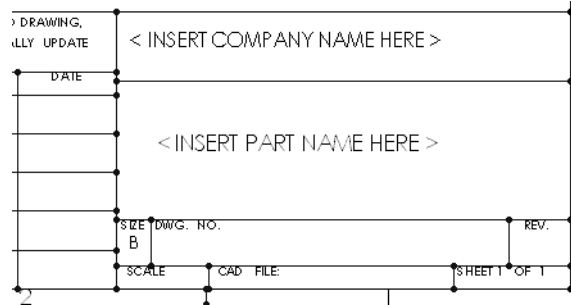
### 3 Views

Create the standard three views of the **Pivot** part in the new drawing.



### 4 Edit the drawing template.

Using **Edit Template**, go to the drawing template. If you haven't already, delete the standard note in the center of the sheet. Zoom in on the title block area.

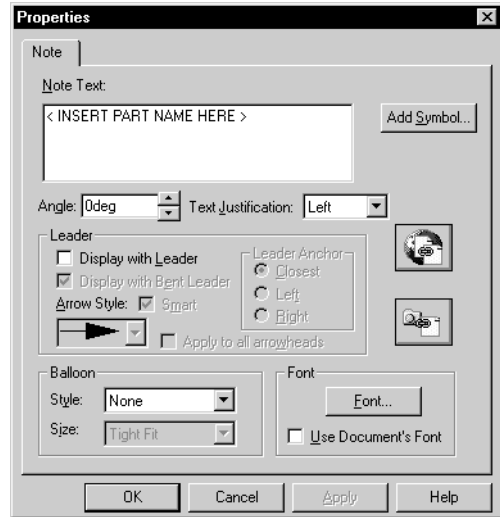


### 5 Edit the note.

The objective is to set up the template so that the name of the part/assembly being detailed will automatically be filled in the appropriate place in the title block.

Double-click on the <INSERT PART NAME HERE> note.

Clear the **Note Text** box by pressing the **Backspace** key.

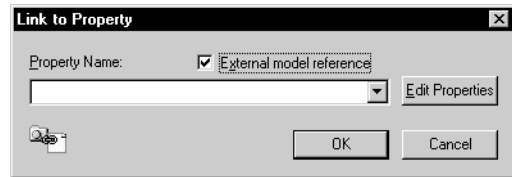


### 6 Link to property.

Click on the **Link to Property** button

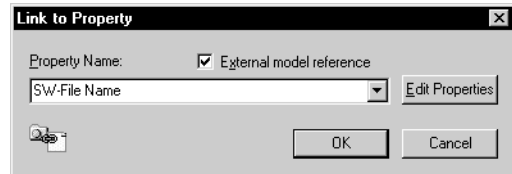


to access the dialog. Properties are normally taken from the active file — the drawing. Click **External model reference** to obtain the properties from the referenced part instead.



### 7 Choose SW-File Name.

The only model property in the list is **SW-File Name**. Click **OK**.

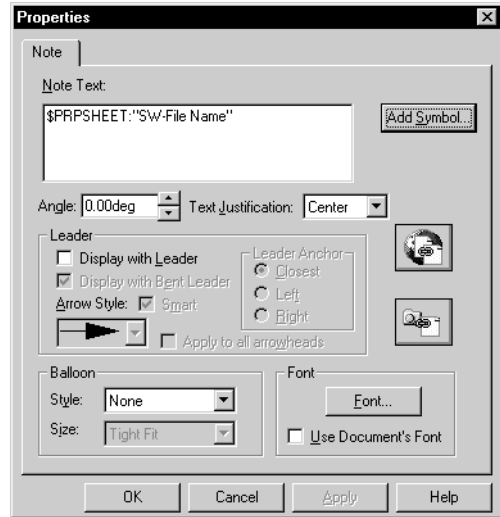


**8 Property.**

The property uses the current part name in the note.

If you wish, change the **Text justification** to **Center** and change the font to something larger.

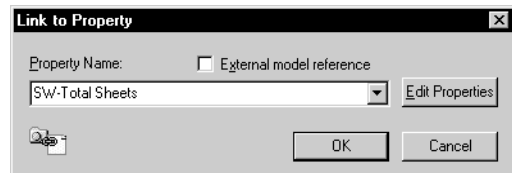
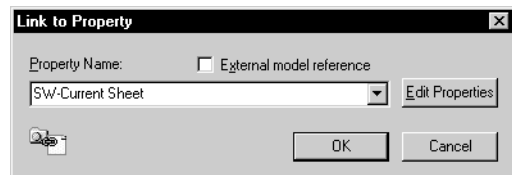
Click **OK**.

**9 Intelligent note.**

The property reads the part name and displays it. If you save the template, these changes will become part of the template and the title block will display the part/assembly name automatically.

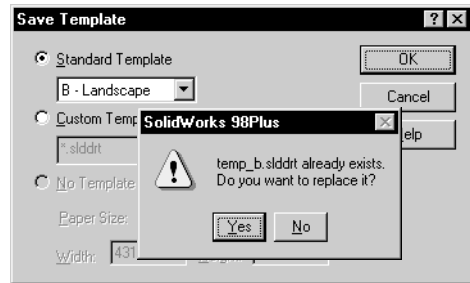
**10 Drawing based properties.**

Edit the four notes that make up the SHEET 1 OF 1 portion of the title block. Change the first "1" in "1 of 1" to use the **SW-Current Sheet** property. For the second "1", use **SW-Total Sheets**.



**11 Save the template.**

Using **File, Save Template** save the changes you have made to the B-Landscape template. If you prefer not to overwrite the standard template, save the changes as a custom template instead.

**12 Close files.**

Close both the part and drawing, saving neither.

**13 Use the template.**

Create a new drawing using the B-Landscape template that you customized. Open the part **Housing** located in the **assy config** folder of the **What's New** directory.

Add the following sheets and views of the part:

**Table 1:**

	Sheet Name	View, Scale
Sheet1	Front	Front, 1:1
Sheet2	Top	Top, 1:1
Sheet3	Right	Right, 1:1

**14 Result.**

Zoom in on the second sheet, **Top**, in the title block area. The part name and sheet number appear in the proper place and are correct.

<i>Housing</i>		
SIZE B	DWG. NO.	REV.
SCALE	CAD FILE:	SHEET 2 OF 2

**15 Close the drawing and part.**

## **Module 5**

# **Assembly Enhancements**





## Edit Subassembly

Sub-assemblies can now be edited in place within the assembly. The action of **Edit Subassembly** is very similar to that of **Edit Part**.

### Valid Operations

There are many operations that can now be performed in the **Edit Subassembly** mode that previously required the sub-assembly to be opened. You can:

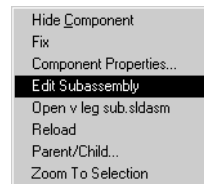
- Add existing components to the sub-assembly
- Add and edit assembly-based features in the sub-assembly
- Delete components from the sub-assembly
- Add new mates to the sub-assembly
- Delete mates from the sub-assembly
- Change the visibility of a sub-assembly component
- Switch configurations of a sub-assembly component

### What You Cannot Do

- **Insert, Component, New**

### Starting Edit Subassembly

Edit a sub-assembly by selecting it and choosing **Edit Subassembly** from the right-mouse menu.



### In the FeatureManager Design Tree

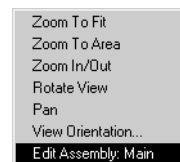
Within the FeatureManager design tree, the sub-assembly being edited appears with colored, rather than black, text. This is consistent with what happens to a part component in **Edit Part** mode.

### On the Screen

The edited sub-assembly appears in the **Edit Part** color. The other components in the assembly appear in their normal colors.

### Returning to Edit Assembly Mode

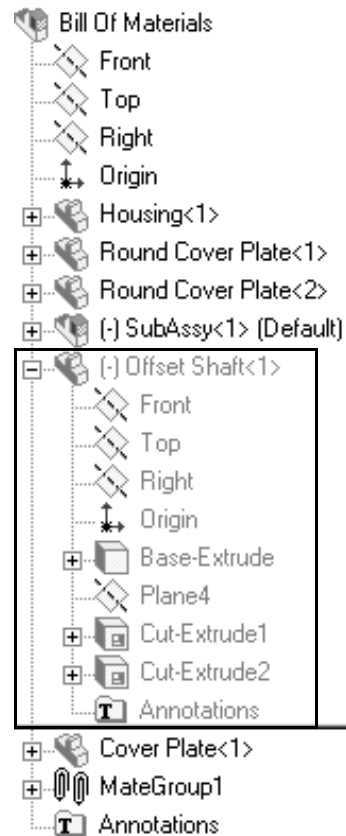
To return from **Edit Subassembly** to **Edit Assembly**, choose **Edit Assembly: assembly name** from the right-mouse menu.



## Edit Part Highlighting

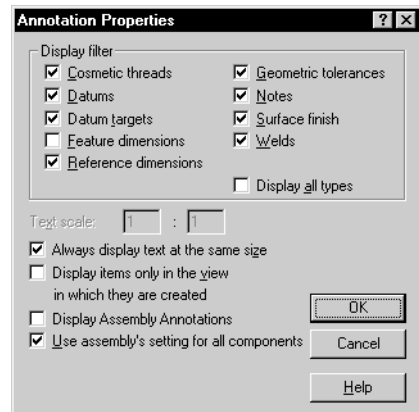
When a part is being edited in the context of an assembly, the part's graphics are highlighted in a salmon color. That has not changed in SolidWorks 98Plus. However, now, the part and its features in the FeatureManager design tree are also highlighted in the same color. You can change the default color through **Tools, Options, Color, Edit Part in Assembly**.

For **Edit Subassembly**, the sub-assembly name and its features are also highlighted in same color.



## Annotations in an Assembly

In SolidWorks98 Plus, you have the option to have the assembly's annotation settings control those of the components too. The **Use assembly's setting for all components** setting is found on the **Properties** dialog of the **Annotations** feature.



## Component Properties

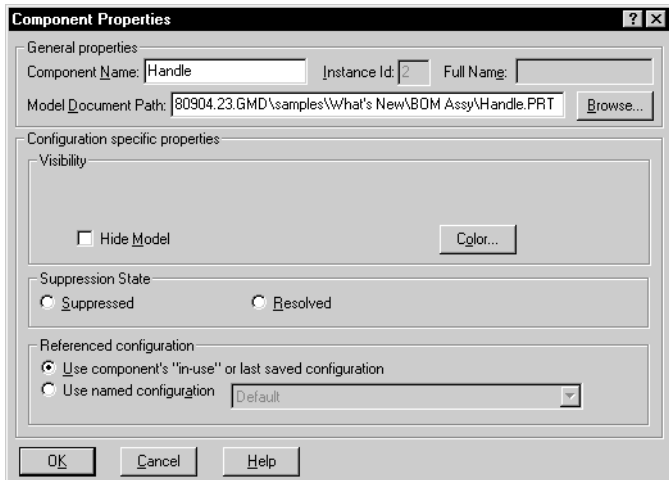
Changes have been made to the **Component Properties** dialog box. The dialog has been rearranged to accommodate the addition of the **Suppression State** options.

The **Visibility** section now contains the **Hide Model** (default is off) option, replacing the **Show Model** checkbox.

The **Color** button has been kept in this area while the **Suppressed** option has been moved.

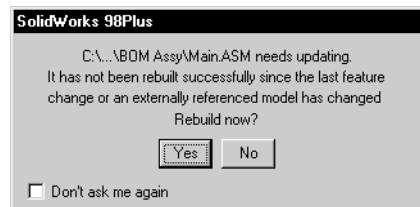
The **General Properties** section now has both **Component Name** and **Fullname** listings. The **Component Name** is editable and can be used to rename the component in the assembly. Changing this name does *not* rename the referenced file.

**Note:** You can't change the name of a component if **Update component names when documents are replaced** is enabled on the **External References** page of the **Options** dialog.



## Update Message

When an assembly is opened and requires updating, a revised dialog appears. The **Don't ask me again** condition remains in effect for the duration of the SolidWorks session.



This message only appears when **Tools, Options, Performance, Rebuild assembly on load** is set to **Prompt**.

## Move Component

It is now possible to switch between components while using **Move Component**. While active in **Move Component**, simply select another component to make it the selected one for move. You can select the component from the screen or the FeatureManager design tree.

You no longer have to pre-select a component before clicking the **Move Component** tool. You can click the tool prior to selecting the component.

## Lightweight Components

To enhance assembly performance in SolidWorks98Plus, components can now be loaded in **Lightweight** status when an assembly is opened. Components that have this **Suppression Status** will be marked appropriately in the FeatureManager or when the cursor is positioned over them in the graphics window.

Lightweight components can:

- Speed up assembly work
- Keep mates intact
- Maintain placement
- Keep orientation
- Be moved and rotated

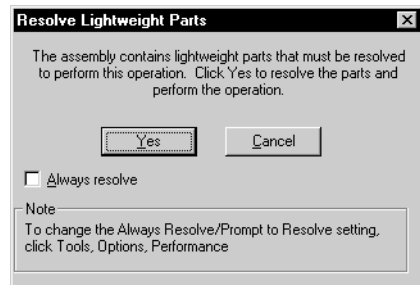
Lightweight components cannot:

- Be edited
- Have their edges, faces or vertices selected or mated to
- Be displayed properly in hidden line or wireframe mode.
- Show its features in the **FeatureManager**
- Have **Mass Properties** or **Interference Detection** done on it
- Be displayed in a drawing


If you try to perform an operation that is not available for lightweight components, you will get a message asking you to resolve the components.

There are now three possible states for a component:

- Resolved (standard state)
- Lightweight
- Suppressed



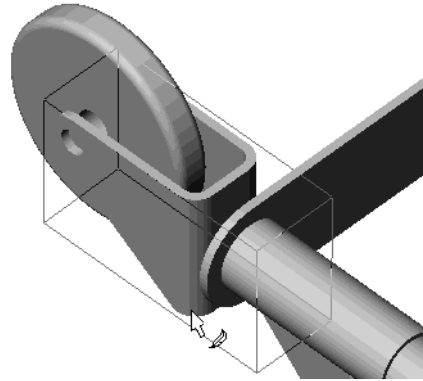
## Appearance

When a component is set to **Lightweight** status, the appearance of the component changes in the FeatureManager design tree. Access to the features of the component are lost and a feather  symbol appears over the component's icon.

The graphic appearance of the component also changes. Cursor feedback when passing over the component changes to the same feather symbol seen in the FeatureManager design tree.

Selecting the component, either in the graphics window or the FeatureManager design tree, will cause the system to resolve the component. That is, the component will be fully loaded into memory and the lightweight designation will be removed.

The FeatureManager listing for a lightweight component cannot be expanded.



## Automatic Mode Setting

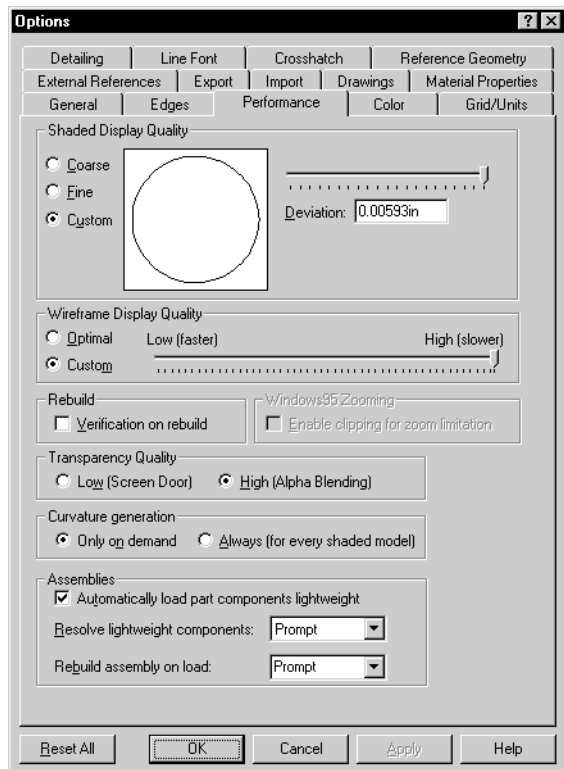
Using a global setting, all part components can be automatically loaded as **Lightweight Components**. The setting affects all assemblies that are opened after the option is set.

Under **Tools, Options, Performance** set the checkbox **Automatically load part components lightweight** to on.

When the setting is on, all component parts in the assembly will be set to lightweight when the assembly is opened. The sub-assemblies will remain **Resolved** so that the components beneath them are visible. However, those components will be loaded as lightweight also.

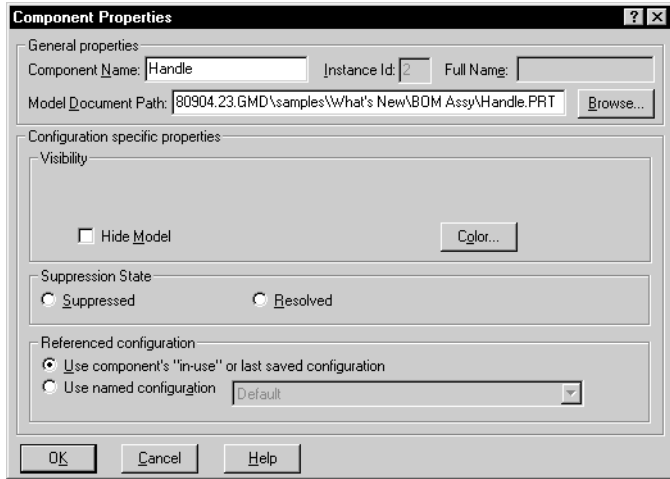
Under certain conditions, the component will change from lightweight to resolved automatically. For example:

- If a lightweight component is selected.
- When an incontext feature is referenced to a lightweight component.
- When equations that affect the lightweight component get solved.



## Changing the Suppression Status

The **Component Properties** dialog has been modified to add the **Suppression State** option. This option consists of two mutually exclusive buttons which set the state of the component. The component can be a part or a sub-assembly.



The same options **Suppress** and **Resolve**, exist on the **Assembly Toolbar**. After selection of one or more components, click on the **Change Suppression State** icon and choose a state from the pulldown menu that appears.



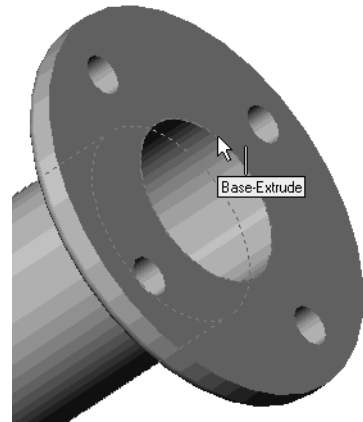
## Assembly FeatureManager

Highlighting of components in an assembly has changed. Select a top-level component, and the component icon highlights. Select a component of a sub-assembly and the sub-assembly icon highlights. If the option **Scroll Selected Item Into View** is toggled on, the highlighted icon will also be scrolled into view.

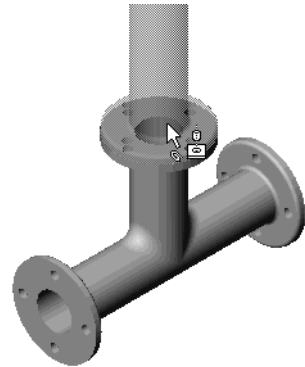
## Smart Mates

A new mating method, **Smart Mates**, can be used to automatically add one or more mates to a component while it is being added to the assembly. To use smart mates, you must drag a feature, edge, vertex or face of an open component part into the active assembly.

In this example, a component is added by selecting an edge and dragging the component into the assembly window.




The geometry you selected is then positioned on corresponding geometry in the active assembly. The component is shown in transparent mode while the mates are previewed. Depending on the types of geometry used, you can create a **Coincident** mate, a **Concentric** mate or both. No mates are created until the component is dropped. That is, until you release the mouse button.





In this example, the edge is dragged near the inner edge of the tee. The cursor feedback indicates that both a **Concentric** and **Coincident** mate will be added if the component is dropped now.

The cursor feedback shows what mates will be added based on the geometry selection.

### ■ Pin in Hole

If you drag a circular edge onto a circular edge (pin in hole), the  cursor is displayed. Both a **Coincident** and **Concentric** mate will be created.

### ■ Planar Faces/Vertices

A planar face dropped onto planar face gives the  cursor and creates a **Coincident** mate. The same mate is created for two vertices .

### ■ Circular Faces

Two circular faces generate a **Concentric** mate and this  cursor.

## Using the Tab Key.

The **Tab** key allows you to switch between various valid solutions. The **Tab** key can be used to:

### ■ Change face orientation

Use the **Tab** key to switch between **Aligned** and **Anti-Aligned**.

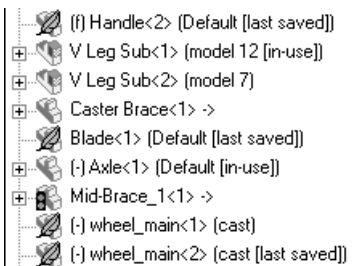
### ■ Rotate a pattern

When two components with circular patterns (such as the pipe flanges shown above) are mated using the pin in hole method, use the **Tab** key to turn the component around the pattern, locking different combinations of pattern features.

## Sub-assembly Configurations

Now each instance of the same sub-assembly can be set to a different configuration in the assembly.

Using the assembly V Leg Sub as an example, the same assembly may contain multiple instances of this sub-assembly. Each instance can now reference a different configuration of that sub-assembly. The configuration name appears at the right of the component name, as it does with parts. Also, a suffix in square brackets indicates whether the configuration is “in use”, or “last saved”.



Configurations of sub-assemblies are created and added in the same fashion as configurations of part components.

## Assembly Statistics

A new option found under **Tools, Assembly Statistics** lists information about the active assembly, components and sub-assemblies.

### ■ Parts

Parts are listed as a total of all part components in the assembly. This includes all instances.

### ■ Unique Parts

This category lists only the first instance of each component part in the assembly.

### ■ Sub-Assemblies

Total of all sub-assemblies in the main assembly.

Assembly Statistics	
Total Number of Components:	25
Parts:	21
Unique Parts:	13
Sub-Assemblies:	4
Unique Sub-Assemblies:	3
Resolved Components:	8
Suppressed Components:	0
Lightweight Components:	17
Number of Top Level Mates:	3
Number of Top Level Components:	9
Maximum Depth:	3
Note: Components of suppressed sub-assemblies are excluded from these statistics.	
OK	



- **Unique Sub-Assemblies**

First instance of each sub-assembly only.

- **Number of Top Level Mates**

Mates in the active assembly, not counting those in any sub-assemblies.

- **Number of Top Level Components**

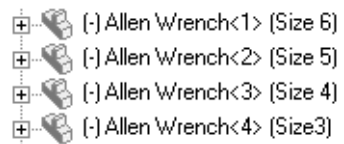
Components listed in the **FeatureManager**, both parts and sub-assemblies. Does not count any components under the sub-assemblies.

- **Maximum Depth**

Depth of nesting levels.

## Part and Sub-assembly Configurations in an Assembly

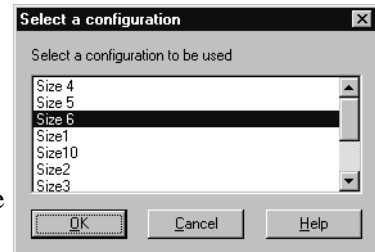
Component parts and sub-assemblies which are in configurations other than the last-saved configuration are now fully displayed and accessible. Each instance has the “+” sign, giving access to all their embedded features.



## Drag and Drop Configurations

Now when you drag and drop a part or assembly with configurations into an assembly from Windows Explorer, you can choose the configuration to use.

Simply select the desired configuration from the **Select a Configuration** dialog, and click **OK**. The last saved configuration is highlighted in the list.



## Assembly Configurations

In SolidWorks 98Plus, more objects can be controlled by a configuration. Now you can control the following aspects of an assembly and have them recognized them as unique to a specific configuration:

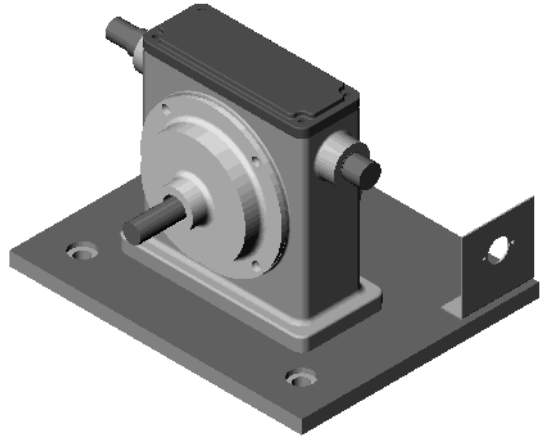
- **Assembly Features**
- **Component Colors**
- **Component Patterns**
- **Cosmetic Threads**
- **Dimensions**
- **Mates**
- **Sketches**
- **Suppression Status**

**1 Open the assembly.**

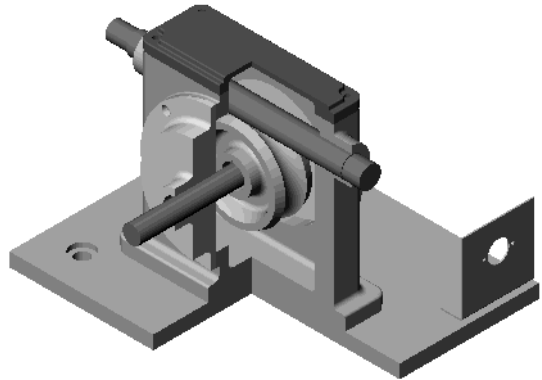
Open the existing assembly MainAssy from the assy config folder in the What's New directory. The assembly consists of several parts and a sub-assembly.

**2 New configuration.**

Add a new configuration named assy section.

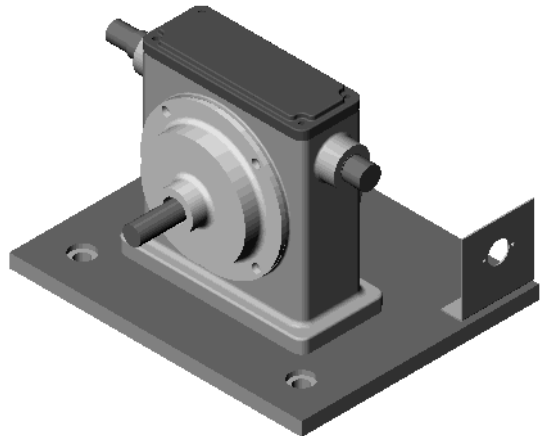
**3 Assembly feature.**

Sketch on the top face of the Cover Plate component. Add an assembly feature with the end condition **Through All**. Edit the **Feature Scope** of the assembly feature to avoid cutting through the Offset Shaft, Worm Gear and Worm Gear Shaft.

**4 Change configurations.**

Return to the Default configuration. The assembly feature does not appear.

This allows you to create a drawing view that references the configuration that shows the assembly cut feature.

**5 Close the assembly**

## Assembly Design Tables

Assemblies can now be controlled by **Design Tables**. These design tables can be used to:

- Set the suppression state of part components
- Set the part components visibility
- Choose a configuration of a part component
- Control assembly features
- Control parameter values for distance and angle mates,
- Add comments to the table

The syntax rules governing design tables have been changed to support their use in assemblies.

- Column headers are STATE, CONFIGURATION, SHOW, PARTNUMBER, NEVER\_EXPAND\_IN\_BOM, USER\_NOTES, and COMMENT.
- The column headers are preceded by the '\$' symbol and are sometimes followed by an '@' symbol and component or feature name.

The format of the design table is the same as in previous versions of SolidWorks. Cells A2, A3, A4, and so on contain the configuration names. Cells B1, C1, D1, and so on contain the dimension name or column tags. The cell A1 remains blank and appropriate values propagate the other cells.

The design table used in an assembly resembles the Excel spreadsheet used in a part. The column headers are always preceded by the '\$' symbol and are sometimes followed by an '@' symbol and component/feature name. The various column headers and their use are explained on the following pages.

## Controlling Part Components

Part components are very versatile in the design table. They can have their configuration, suppression state and visibility set through the design table. It is important to note that part and configuration names are case sensitive.

D	E	F
\$CONFIGURATION@part_a<1>	\$STATE@Part_b<*>	\$SHOW@partC<3,4,7>
Default	suppressed	yes
Closed	RESOLVED	no
open	resolved	No

### ■ Configurations

To control the configuration of a part component use the column header name CONFIGURATION followed by the part name and instance. A typical use would be \$CONFIGURATION@part\_a<1> where part\_a is the part name and <1> is the instance number of the component that is affected.

The cells beneath the column header must contain valid configuration names for that part.

#### ■ State

To control the suppression state of a part component, use the column header **STATE**. The statement `$STATE@Part_b<*>` is used to set the state for *all* instances of the part component `Part_b`.

The cells beneath this type of header can contain suppressed (abbreviated *s*) or resolved (abbreviated *r*).

#### ■ Visibility

A part component's visibility can be set using the **SHOW** header. The syntax is `$SHOW@partC<3,4,7>`, this sets the visibility for instances 3, 4 and 7 of the part component `partC`. Note that a range of instances `<3-8>` can be used alone or in conjunction with a set of instances like this `<1,4-7,12>`.

The cells beneath this type of header have either yes (abbreviate *y*) to **Show** or no (abbreviate *n*) to **Hide** them.

## Specifying Components

Any design table function that allows you to specify component names supports the following syntax. In this list, name is the component name and n is the instance number.

- name<n> .....Affects only the specific instance of name.
- name<\*> .....Affects all instances of name.
- name<n-m>.....Affects all instances of name within the range of n-m.
- name<n,m>.....Affects only the listed instances of name.

## Assembly Features and Mates

Assembly features can be suppressed or unsuppressed using the design table. In addition, dimensions from the assembly feature or distance/angle mates can be set to different values.

G	H
<code>\$STATE@assy-feat1</code>	<code>DI@Distance1</code>
suppressed	10
Unsuppressed	12
U	12

#### ■ State

The suppression state of an assembly feature can be specified by configuration. Using the **STATE** header, the statement `$STATE@assy_feat1` is used to set the state for the assembly feature `assy_feat1`.

The cells beneath this type of header can have suppressed (abbreviate *s*) or unsuppressed (abbreviate *u*) listed.

### ■ Dimension

The value of a dimension in the assembly, either a distance or angle mate or a dimension in an assembly feature, can be used as a column header. Use the **FullName** of the dimension from **Dimension Properties**. The distance mate has the form D1@Distance1. The angle mate has the form D1@Angle1. Assembly features uses standard sketch naming such as D3@Sketch5.

### Comments and Other Headers

Several special column (and row) headers now exist for comments and BOM settings. They are:

#### ■ Comments

Using the `COMMENT` header, you can enter text into the cells of a column and it will be ignored by SolidWorks. The syntax for the column header is `$COMMENT`.

#### ■ Part Number

The `PARTNUMBER` header is treated the same as the `COMMENT` header. The syntax is `$PARTNUMBER` and offers a more descriptive column name.

#### ■ User Notes

The comment can be used as either a column or row header. The syntax is `$USER_NOTES`.

#### ■ BOM Expansion

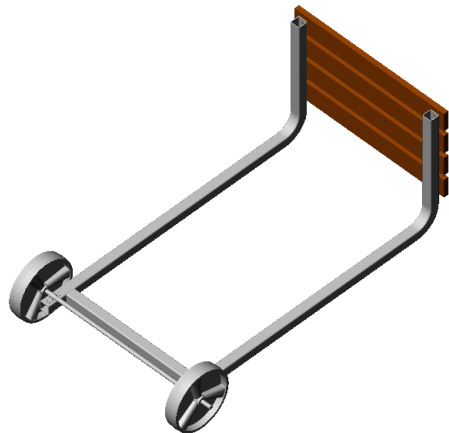
This header allows you to prevent the display of sub-assembly child components in the bill of materials. The header syntax is `$NEVER_EXPAND_IN_BOM` and you put either a *yes* (abbreviate *y*) or *no* (abbreviate *n*) in the cells below it.

### 1 Open the assembly.

Open the existing assembly `SUBASSY_LEGS` from the `assy_dt` folder in the *What's New* directory.

### Part Configurations

One of the parts in the assembly requires two configurations. The part will be opened and edited. Note that the same result could be achieved using a design table.



**2 LEG component.**

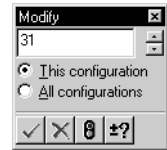
Open the LEG component from the assembly. In this part, add two new configurations, HOLES and NO\_HOLES.

The HOLES configuration is the same as the current Default configuration.

In the NO\_HOLES configuration suppress the feature BRACKET\_HOLES.

**3 Change value by configuration.**

Switch to the NO\_HOLES configuration. Double-click on the Base feature and then on the **29in** dimension. A dialog appears asking whether the change is to be applied to the current (NO\_HOLES) or all configurations. Set the value to **31** for **This configuration** and rebuild. The value will differ between configurations without the use of a design table.

**4 Comparisons.**

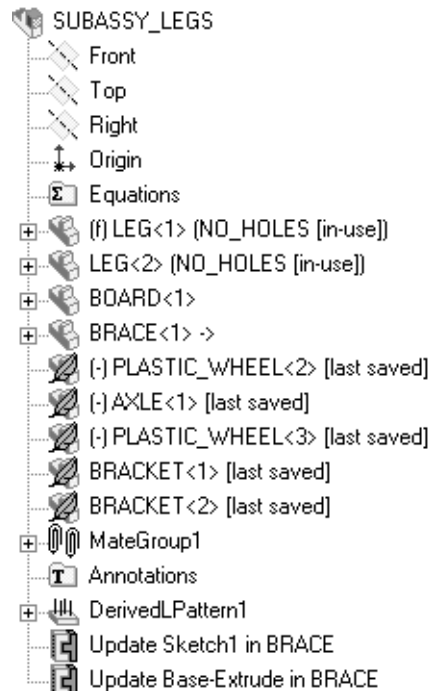
The resulting configurations are the same as those arrived at in the design table exercise in the *Module 3: Modeling Enhancements*.

**5 Close and Save the file.**

The assembly will be updated as the file closes.

## 6 FeatureManager design tree.

In the FeatureManager design tree, the LEG components now display the configurations that they are using. The **Last Saved** configuration is always used unless otherwise specified.



## Assembly Design Tables

There are two versions of this assembly which will be used in the same main assembly. To create these configurations, a design table in the assembly will be used.

## 7 Spreadsheet.

Open a new Excel spreadsheet and add the information shown below. Copy the component names (BRACKET, WHEEL and AXLE) from the **Component Properties** dialog. The two new configurations will be WHEELS and NO\_WHEELS.

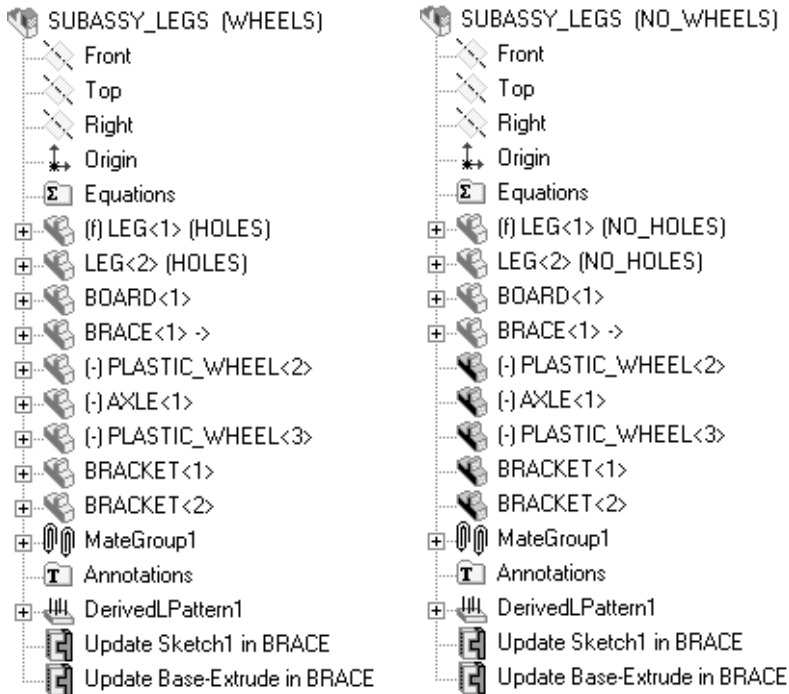
	A	B	C	D	E
1		\$CONFIGURATION@LEG<*>	\$STATE@BRACKET<1,2>	\$STATE@PLASTIC_WHEEL<*>	\$STATE@AXLE<1>
2	WHEELS	HOLES	RESOLVED	R	RESOLVED
3	NO_WHEELS	NO_HOLES	SUPPRESSED	S	SUPPRESSED
4					
5	USER_NOTES		several instances	all instances	one instance

## 8 Insert design table.

After the design table is inserted, the top level component lists the current configuration.

The WHEELS configuration has all components **Resolved**.

The NO\_WHEELS configuration has all components **Resolved** except for all instances of the PLASTIC\_WHEEL, BRACKET and AXLE, which are suppressed.



## 9 Set the configuration.

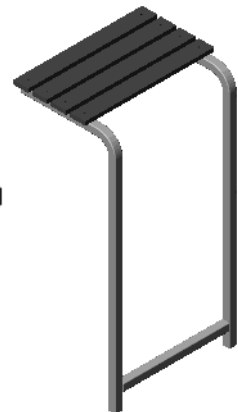
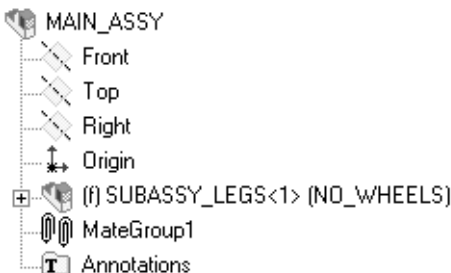
Change to the NO\_WHEELS configuration and save the assembly file.

## 10 Main assembly.

Open a new assembly with units set to **inches**. This will act as the main assembly for two instances of the SUBASSY\_LEGS assembly. Name it MAIN\_ASSY.

## 11 Insert sub-assembly.

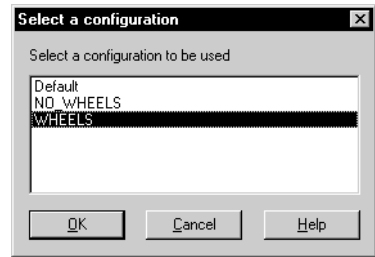
Using **Insert, Component, From File**, insert SUBASSY\_LEGS at the assembly origin. Since the sub-assembly is added with the last saved configuration active, it comes in with its configuration set to NO\_WHEELS.



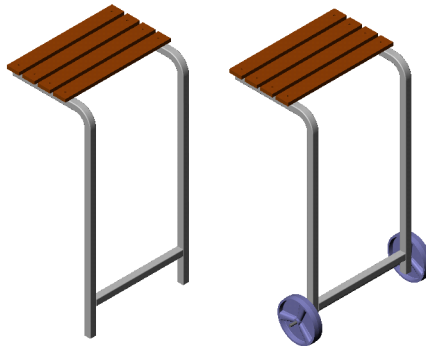
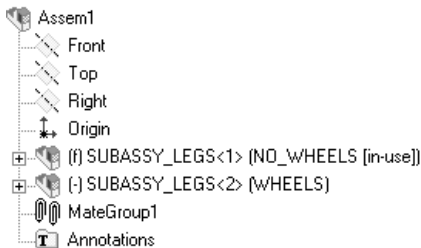


**12 Explorer.**

Open Windows Explorer and find the file SUBASSY\_LEGS.sldasm. Drag and drop the file into MAIN\_ASSY. A dialog listing the possible configurations is displayed. Choose WHEELS and click **OK**.

**13 Second instance.**

The second instance of SUBASSY\_LEGS is dropped into the MAIN\_ASSY. The dialog enables you to choose the desired configuration on the fly.

**14 Mating.**

Mate the second instance to the first as shown. The distance is measured between the facing legs.

**15 Close all the files.**

