SolidWorks 2000 CreoScitex Training Level 2

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Level 2

Audience: Proficient SolidWorks users

Level 2 training is for the average SolidWorks user looking to become proficient in the details of SolidWorks design, documenting and file management techniques.

Prerequisites: 1) Completion of Level 1 training and several (full) days of SolidWorks usage OR 2) equivalent experience using SolidWorks on the job at Creo

Course Objectives

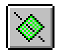
By the end of the Level 2 two day course, the student will be able to:

- Work within the CreoScitex drafting standards.
- Use the CreoScitex title blocks and note blocks.
- Manage part and assembly files.
- Use configurations for motion – e.g. and extended cylinder and retracted cylinder with resulting motion.
- Create a "simplified" configuration of a part, subassembly & assembly
- Install/use the pallet parts and part libraries
- Add features to the "common" feature pallet.
- Create complete notes on the Model for CNC.
- Use weld symbols
- Create tolerance parts - GD&T
- Work with SolidWorks explorer to:
    - Preview parts and assemblies
    - Edit parts, sub assemblies or assemblies
    - Change parts in an assembly – Replace, Rename, Copy
    - Edit configurations
    - Show references
    - Where used.
- Understand the revision process.
- Redefine Views
- Correct common constraint errors.
- Set Detailing Drawing Options
- Use CreoScitex standard Text and Dimensions.
- Use Text and Dimension style overrides.
- Create a standard 3view drawing from part or Assemblies.
- Create part Balloons
- Create a Bill of Materials (suitable for use in SAP)
- Create a Sheet metal part including:
    - Bending Reliefs
    - Standard "pen nut" features.
    - Thin features.
    - "Rollback-check"
- Work with advanced hole wizard options.
- Use the feature history tree to make appropriate part changes.
- Import Export File formats (IGES, Unigraphics, DXF)
- Use rib features.
- Use parting (split part).
- Use draft.

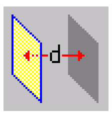
Plane Overview

You can create planes in a part or assembly. You can use planes to sketch, to create a section view of a model, for a neutral plane in a draft feature, and so on.

To create a construction plane:

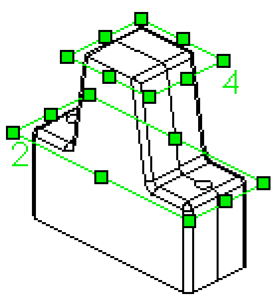
1. Click **Plane**  on the Reference Geometry toolbar, or click **Insert**, **Reference Geometry**, **Plane**.
2. Select the type of plane you want to create and click **Next**.

Offset Plane

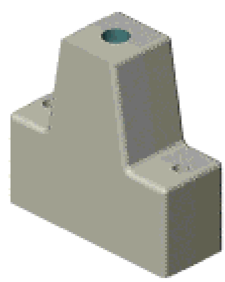


To create a plane parallel to a plane or face, offset by a specified distance:

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

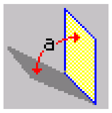


Plane 4 is offset from Plane 2. Because the model is drafted, a plane must be created for the sketch of a hole.



The hole is cut in the top of the model.

Plane at Angle



To create a plane through an edge, axis, or sketch line at an angle to a face or plane:

1 Select a plane or a planar face.

2 Select an edge, axis, or sketch line.

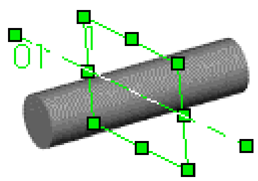
3 Specify the **Angle** between the new and existing plane.

4 Select the **Reverse Direction** check box, if necessary.

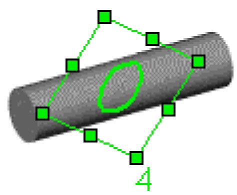
5 Click **Finish**.

The behavior of the plane is as follows:

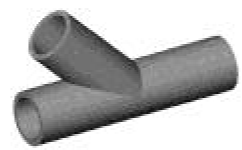
- If the selected line is in the same plane as the selected plane, the new plane rotates around the selected line.
- If the selected line is parallel to the selected plane, the new plane moves to the parallel line and rotates around the line.
- If the selected line is skewed with respect to the selected plane, the selected line is projected onto the selected plane and then the plane rotates around the projected line.



Use Plane 1 and Axis 01 to create a new plane at an angle.

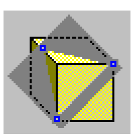


Sketch the extrusion profile on Plane 4.



Create the Y-shaped pipe.

Three Point Plane

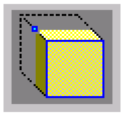


To create a plane through three points:

1 Select three vertices, points, or midpoints.

2 Click **Finish**.

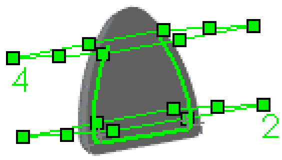
Parallel Plane at Point



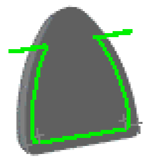
To create a plane through a point parallel to a plane or face:

1 Select a plane or planar face and a vertex, point, or midpoint.

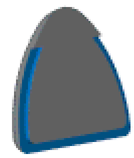
2 Click **Finish**.



Plane 4 is parallel to Plane 2 at the endpoint of the guide curve sketch.

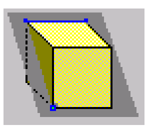


The two loft profiles are sketched on Plane 4.



The lofted cut (shown in blue) is added to the model.

Line and Point Plane

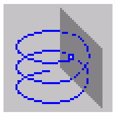


To create a plane through an edge, axis, or sketch line and a point:

1 Select an edge, axis or sketch line and a vertex, a point, or a midpoint.

2 Click **Finish**.

Perpendicular to Curve at Point Plane

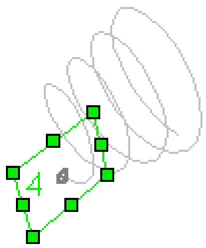


To create a plane through a point and perpendicular to an edge, axis, or curve:

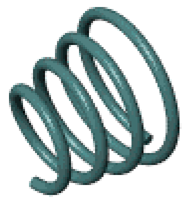
1 Select an edge, axis, or curve and a vertex or point.

2 Select the **Set Origin on Curve** check box to place the origin on the curve; the default is to place the origin on the vertex or point.

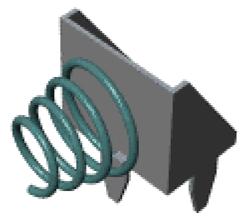
3 Click **Finish**.



Plane 4 is perpendicular to the end of the tapered helix.



A circle is swept along the helix to create a spring.



The helical spring is used in a battery contact terminal.

File Locations

Lets you specify and locate referenced documents. Folders are searched in the order in which they are listed in the **Show folders for** list.

To display File Locations:

1. Click **Tools, Options, System Options.**
2. On the **System Options** tab, click **File Locations**.

The **Show folders for** box displays search paths for files of various types, including:

- Referenced Documents
- Palette features
- Palette parts
- Palette forming tools
- Blocks
- Document Templates
- Sheet Format

If one or more folders are displayed in the **Folders** list, you can **Add** new folders, **Delete** existing folders, and **Move Up** or **Move Down** to change the search order.

To add a folder:

1. Select a folder type from the **Show folder for** list.
2. Click **Add** to add a new directory path to the list
3. Browse in the **Choose Directory** dialog box to locate the folder.
4. Click **OK** to add the folder to the **Folders** list.

To delete a folder:

To delete a directory path from the list, select the path in **Folders** and click **Delete**.

To move a folder:

To change the order of the list, select the path in **Folders** and click **Move Up** or **Move Down** as needed.

NOTE: The paths for Referenced Documents are searched only if **Search document folder list** for external references is selected, in **System Options, External References**.

The paths for Palette parts, Palette features, Palette forming tools, and Blocks are always searched, whether **Search document folder list** for external references is selected or not.

Palette Features and Library Features - Similarities and Differences

Similarities between palette features and library features:

- You create a palette feature in the same way as a library feature.
- The palette feature is displayed as a library feature in the FeatureManager design tree of the target part.
- You can **dissolve** a palette feature the same way as a library feature.

Differences between palette features and library features:

- When you add a palette feature to a part, you can simply drag the feature roughly where you want it to be, then place it precisely by re-attaching and modifying any locating dimensions saved in the feature.
- You can edit the dimensions of the palette feature as you insert it into the part.
- Not all library features are suitable for use in the Feature Palette window. A palette feature is limited to *one* mandatory reference (one face, plane, edge, or vertex) on the target part. See **Mandatory References** .
- Library features that have multiple mandatory references (such as draft features that depend on the selection of several faces) cannot be added through the Feature Palette window.

You can also use the Feature Palette window to add parts to an assembly. You can organize parts in logical groups, and you can see a miniature graphical view of each palette part.

You can add fittings to piping assemblies by dragging them from the Feature Palette window. For more information, see SolidWorks Piping Help Topics.

The Feature Palette window is also used to apply forming tools to sheet metal parts. Forming tools are special parts that punch, stretch, or shape the sheet metal part, to create embossments, lances, flanges, louvers, ribs, and so on.

Mandatory References

A palette feature is limited to *one* mandatory reference on the target part.

The limitation of a single mandatory reference does not mean that a palette feature can consist of only one feature. As long as the *parent* feature uses a single reference, the palette feature can include multiple *child* features. To examine the parent/child relationships, right-click the feature, and select **Parent/Child**.

To determine if a library feature is suitable for use in the Feature Palette, insert it as a library feature in a target part (see **Adding a Library Feature to a Part**). If there is only one mandatory reference listed, you can use the library feature as a palette feature.

Feature Limitations

Because of the limitation of a single reference, lofts, sweeps, and shape features are allowed only in certain cases:

- For lofts, the single mandatory reference is the sketch plane of one of the profiles. This must be the face or plane where you drop the feature. Additional sketch planes must be offset from this plane, or from each other. The library feature must include the planes.
- For sweeps, the sweep *path* is the single mandatory reference. This must be a model edge. The sketch plane for the sweep *section* must be a **Perpendicular to Curve** plane, with the model edge as the curve (when you select the edge, click near the vertex where the plane should be placed). Include this plane in the library feature. Then, when you drop the palette feature in the target part, drop it on the edge that is the sweep path, near the appropriate vertex.
- For shape features, the shape must not be constrained to any entities. The definition can include only variations in **Pressure**, **Bend**, and **Stretch** (on the **Controls** tab of the **Shape Feature** dialog box).

Add Configuration/Configuration Properties

1 Enter a name for the configuration in the **Configuration Name** box. The name must not include the forward slash (/) or *at* (@) character. A warning message appears when you close the dialog box if the name field contains either of these characters, if the field is blank, or if the name already exists.

2 Enter a **Comment** that describes the configuration. (Optional)

3 Select **Properties for newly inserted items**.

These options control what happens when new items are added to another configuration, and then this configuration is activated again. They apply only to assembly configurations:

- **Suppress features and mates.** When checked, new mates and features added to other configurations are suppressed in this configuration. Otherwise, new mates and features are unsuppressed in this configuration also.
- **Suppress component models.** When checked, new components added to other configurations are suppressed in this configuration. Otherwise, new components are resolved in this configuration also.
- **Hide component models.** When checked, new components added to other configurations are hidden in this configuration. Otherwise, new components are shown in this configuration also.

This item applies only to part configurations:

- **Suppress Features.** By default, newly added features are unsuppressed in the active configuration. This option controls what happens when new features are added to another configuration, then this configuration is activated again.

  When checked, new features added to other configurations are suppressed in this configuration. Otherwise, new features are unsuppressed in this configuration also.

4 Select the way properties are applied (only for assemblies):

- **Apply properties to sub-assembly root only.** When selected, the properties chosen above apply to the sub-assembly root only.
- **Apply properties to ALL sub-assembly components.** When selected, the properties chosen above apply to all components in the sub-assembly. (Use this option when you want to hide or suppress most of the components in a large assembly. You can then individually select the components that you want unsuppressed or visible.)

5 Click **Advanced** to specify how the assembly or part is listed in a Bill of Materials.

6 Click **Custom** to access the **Summary Info** dialog box.

7 New! To specify a color for the configuration, select the **Use configuration specific color** check box, then click **Color** to specify a color from the color palette for the configuration.

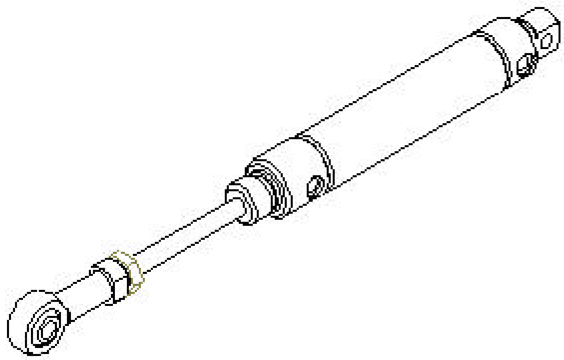
If the color for wireframe and HLR modes is the same as the color for shaded mode, the configuration specific color applies to all three modes. If the color is not the same for all three modes, then the configuration specific color applies to shaded mode only. To be sure the colors are the same for the three modes, select the **Apply same color to wireframe, HLR and shaded** check box in **Tools, Options, Document Properties, Colors**.

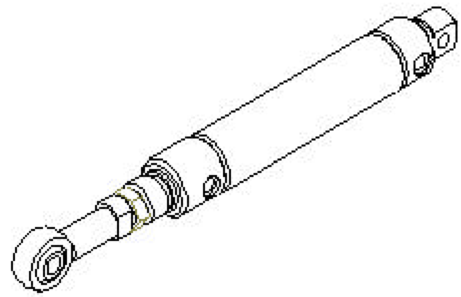
8 Click **OK.** The new configuration name appears in the tree.

9 Change the model as needed to create the design variation.

- In a part, you can suppress features, modify dimensions, add custom properties, and so on.
- In an assembly, you can suppress or hide components, choose a different referenced configuration of components, and so on.

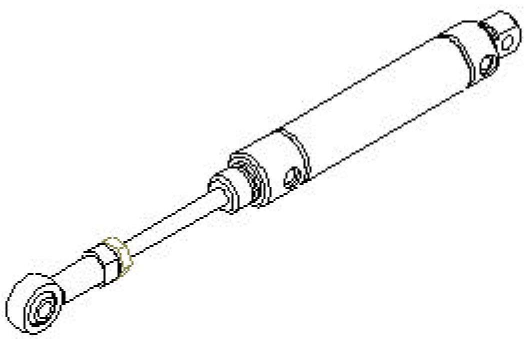
Configuration example – extend & retract cylinder

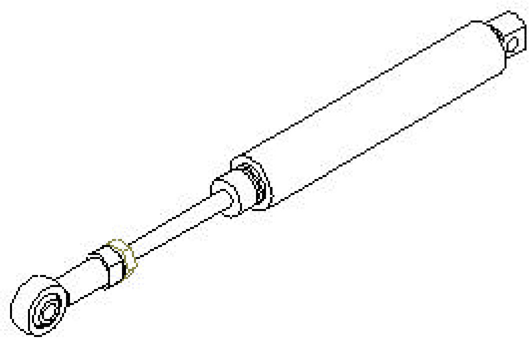


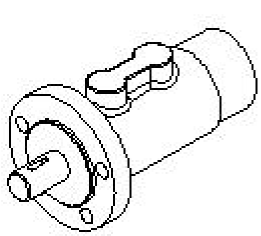


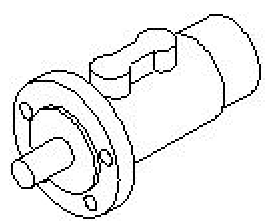
\Example Files\Cyl**990625 cyl 5625 150 w rod end.SLDASM**

Configuration examples – detailed & "simplified"









Suppress and Unsuppress Features

To suppress a feature:

1 Select the feature in the FeatureManager design tree, or select a face of the feature in the graphics area. To select multiple features, hold down **Ctrl** as you select.

2 Click **Suppress** on the Features toolbar, or click **Edit, Suppress.**

- or -

1 In the FeatureManager design tree, right-click a feature, and select **Properties**.

2 In the **Feature Properties** dialog box, select the **Suppressed** check box, and click **OK**.

The selected feature is removed from the model (but not deleted). The feature disappears from the model view and is shown in gray in the FeatureManager design tree.

To unsuppress a feature:

1 Select the suppressed feature in the FeatureManager design tree.

2 Click **Unsuppress** on the Features toolbar, or click **Edit, Unsuppress**.

- or -

1 In the FeatureManager design tree, right-click a feature, and select **Properties**.

2 In the **Feature Properties** dialog box, clear the **Suppressed** check box, and click **OK**.

If you want to unsuppress a suppressed feature, you must select it using the FeatureManager design tree.

To unsuppress a feature and its dependents:

1 Select the suppressed parent feature in the FeatureManager design tree.

2 Click **Unsuppress with Dependents** on the Features toolbar, or click **Edit, Unsuppress with Dependents**.

The selected feature and any features that are dependent on it are returned to the model.

- or -

1 Select the child feature in the FeatureManager design tree.

2 Click **Unsuppress with Dependents** on the Features toolbar, or click **Edit, Unsuppress with Dependents**.

The selected feature and its parent feature are returned to the model.

Managing part and assembly files.

Simplifying Large Assemblies

Large assemblies can consist of hundreds of components. You can simplify a complex assembly by toggling the **visibility of the components**, and by changing the **suppression state of components**.

There are several reasons to simplify an assembly:

- To improve system performance, and to reduce rebuild times, especially with very large assemblies.
- To create simplified views of the assembly that include certain components, and exclude others.
- To create design variations of the assembly, with different combinations of components, and different configurations of the components themselves.

Lightweight Parts

You can load an assembly with its active parts *fully resolved* or *lightweight*.

- When a part is *fully resolved*, all its model data is loaded in memory.
- When a part is *lightweight*, only a subset of its model data is loaded in memory. The remaining model data is loaded on an as-needed basis.

You can improve performance of large assemblies significantly by using lightweight parts. Loading an assembly with lightweight parts is faster than loading the same assembly with fully resolved parts.

Lightweight parts are efficient because the full model data for the parts is loaded only as it is needed. Only parts that you select, and parts that are affected by changes that you make in the current editing session, become fully resolved.

Assemblies with lightweight parts rebuild faster because less data is evaluated. Mates on a lightweight part are solved, and you can edit existing mates.

To enable lightweight loading of parts:

1. Click **Tools**, **Options**. On the **System Options** tab, click **Performance**.
2. Under **Assemblies**, click the **Automatically load parts lightweight** check box.

NOTE: You can only change this option when no assemblies, or drawings of assemblies, are open.

When this option is selected, all parts are loaded lightweight when you open an assembly. The only exception is that any parts that are included in the feature scope of assembly feature are always loaded fully resolved. Sub-assemblies themselves are not loaded lightweight, but the individual parts that they contain are lightweight.

When you point at a lightweight part in the graphics area, a feather pointer  is displayed, and the body is surrounded by a bounding box.

NOTE: Lightweight parts are only used when the model is in **Shaded** mode. If the model was last saved in **Hidden Lines Removed**, **Hidden In Gray**, or **Wireframe** mode, lightweight parts are not used.

Assembly Features

While in an assembly, you can create cut or hole features that exist in the assembly only. You determine which parts you want the feature to affect by setting the scope. New! You can create a pattern of assembly features in the same manner as you create a pattern of features in a part.

This is useful for creating cuts or holes that are added after the components are actually assembled, and that affect more than one component. When you want to add a cut or hole to a single component in an assembly, it is better to edit the part in context than to use an assembly feature.

While it is not a requirement, it is good practice to fully define the positions of the components of the assembly, or fix their locations before you add assembly features. This helps prevent unexpected results if the components are moved later.

Replacing a Single Instance of an Assembly Component

You can replace only the selected instance of a component.

To replace a single instance of a component by editing its properties:

1. In the FeatureManager design tree or the graphics area, right-click the component you want to replace, and select **Component Properties**.
2. Enter a **Component Name** if you want the FeatureManager design tree to list the component by a name that is different from the name of the component file.

   If the **Component Name** box is unavailable, click **Tools**, **Options**. On the **System Options** tab, under **External References**, clear the **Update component names when documents are replaced** check box.

   The **Component Name** is used *in the FeatureManager design tree only*; the component's file name does not change. Component names may not include the forward slash (/) or at (@) characters.
3. To locate the replacement component, click **Browse**. Navigate to the new document. If desired, click the **Preview** check box to view the component in the **Preview** window.
4. Click **Open**. The **Model Document Path** displays the selected document name.
5. Click **OK** to replace the component.

   The last saved or in-use configuration of the selected component is used.

Preserving Mates with Component Replacement

When replacing a component, mates used in the original part are applied to the replacement part wherever possible. Preservation of mates depends on the entities used in the mates being the same. If the entities used in the mates are not the same, or do not exist, in the replacement part, the mates will have an error.

You can rename the corresponding edges and/or faces on a replacement part to match the edge/face names on the original part to ensure that the mates are preserved.

To manually name the entities used in a mate:

1. In the *original* part document, right-click the entity used in the mate, and select **Face Properties** (or **Edge Properties**).
2. Under **Entity Information**, enter a new **Name**, and click **OK**.
3. In the replacement part document, right-click the corresponding edge or face, and edit the entity properties, using the same **Name**.

   TIP: To automatically name the entities used in a mate, click the **Automatically generate names for referenced geometry** check box on the **External References Options** dialog box.

To identify named entities:

1. In the part document, right-click the part icon at the top of the FeatureManager design tree, and select **List Named Entities** (if the part has no named entities, the shortcut menu does not include this selection.)
2. To highlight a named entity, click the item in the **Named Entities** dialog box.

What's Wrong?

Lets you view any rebuild errors of a part or assembly. A down arrow appears next to the name of the part or assembly, and the name of the failed feature. An exclamation mark indicates the item responsible for the error.

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

Some common rebuild errors include:

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

If an error message has a prefix of **, you can click on the error message and the problem area is highlighted on the model.

You can turn off the automatic display of errors by clearing the **Display Errors at Every Rebuild** check box.

NOTE: The **Display Errors at Every Rebuild** check box affects only the current session.

Select the **Display Full message** check box if you want the complete message displayed whenever there is an error. Otherwise, an abbreviated message is displayed. (The default is **Display Full message**.)

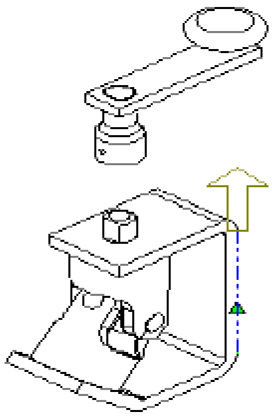
The **Rebuild Errors** dialog box is displayed when the error is first generated, or you can display the dialog box at any time by right-clicking on the part in the FeatureManager design tree.

Alternative Method for Editing an Exploded View

To edit an exploded view without the Assembly Exploder dialog box:

1. Double-click the **ExplView** feature in the ConfigurationManager to explode the view.
2. Right-click the component you want to reposition, and select **Show Explode Steps**. You can right-click the component in the graphics area, or you can switch to the FeatureManager, then right-click the component in the tree.

   The drag handles for the explode steps on the selected component are displayed.

3. Drag the component by the green drag handle to the new position.
4. Repeat as needed for each component, then collapse the assembly.

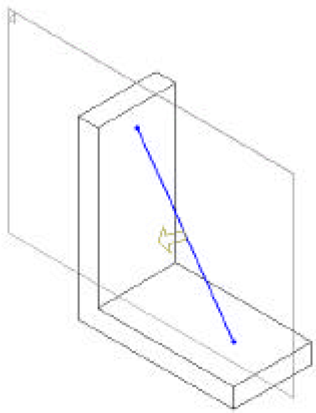
Rib Feature

You can create a rib using either single or multiple open, or closed sketches.

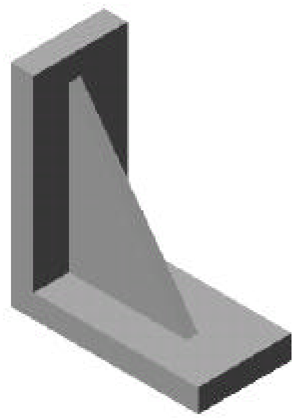
To add a rib using either an open or closed sketch:

1 Using a plane that intersects the part, sketch the contour to be used for the rib.

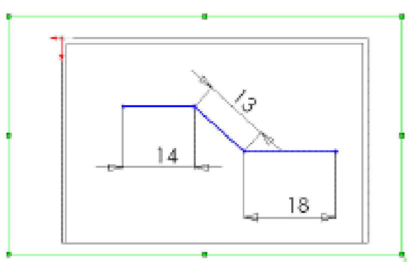
You can sketch a single, or **multiple elements**, using either closed or open contours, to create the rib.



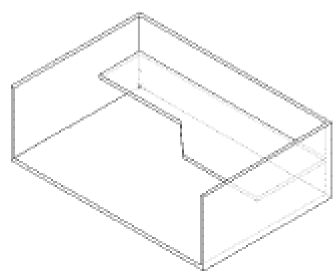
Sketch for rib feature



Rib applied selecting **Parallel to sketch**

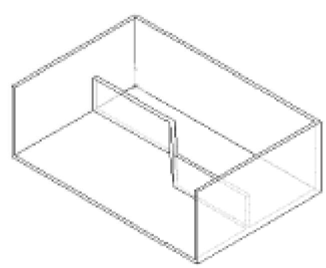


Sketch for rib using open sketch elements



Rib applied selecting **Parallel to sketch**

Rib using open sketch elements, selecting **Normal to sketch**



2 Click **Rib** on the Features toolbar or click **Insert, Features, Rib**.

In the PropertyManager, set the following values:

1 For **Thickness**, select **Mid plane** to extrude the rib equally in both directions from the sketch plane, or **Single side** to extrude in one direction.

If you choose **Single side**, examine the preview and select **Reverse** if necessary to get the results you want.

2 Enter the thickness of the rib, or click the arrows to change the value.

3 For **Extrusion Direction**, select **Parallel to sketch** to extend parallel to the sketch, along the plane, or **Normal to sketch** to follow the direction of the sketch.

Note the direction of the preview arrow, and click the **Flip material side** check box, if necessary.

4 For **Extension Type**, select **Natural** to follow the curve in the sketch of the rib or **Linear**, to extend in a linear fashion from the sketch of the rib.

NOTE: If you select **Parallel to sketch** as the **Extrusion Direction**, you can only select **Natural** as an **Extension Type.**

5 To add a draft, select **Enable draft** in **Draft conditions**, and enter the draft **Angle** or click the arrows to change the value.

6 Select **Draft outward** if necessary.

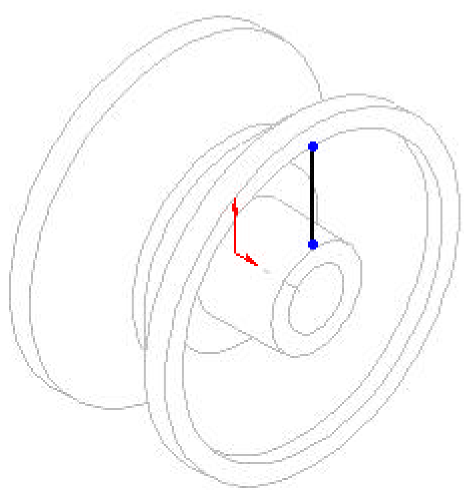
7 To create a rib with multiple drafts using draft angles, click **Next Reference** until the arrow shows on the contour from which you want to start the draft angle.

Enter a draft angle for the first contour, and click **Next Reference** to proceed to the next contour.

8 Repeat until you have selected all the contours and applied a draft **Angle** to each.

9 Click **OK**.

Add Rib Feature Example



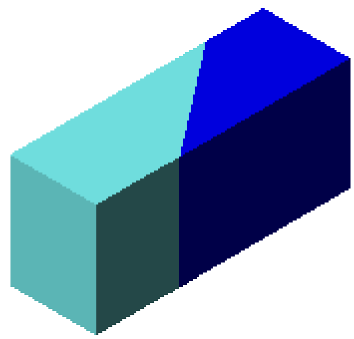
\Example Files\Vroller**Vroller.SLDPRT**

Parting Line Draft

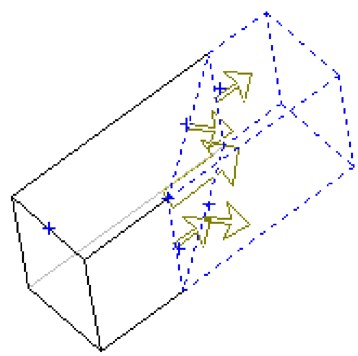
The parting line option lets you draft surfaces around a parting line. The parting line can be non-planar. In a draft using a parting line, you can also include a **Step draft**.

To draft on a parting line, you may first divide the faces to be drafted by inserting a **Split Line**, or you may use an existing model edge. Then you specify the direction of pull, or the side of the parting line from which material is removed.

To insert a draft angle parting line:



1. Sketch the part to be drafted.
2. Insert a split line curve, if desired.
3. Click **Draft**  on the Features toolbar or click **Insert, Features, Draft**.
4. In the **Type of draft** box, select **Parting Line**.
5. Enter the **Draft angle**.



6. Click the **Direction of Pull** box, and select an edge or face in the graphics area to indicate the direction of pull.

   Note the arrow direction, and click **Reverse direction** if desired.
7. Click the **Parting lines** box, and select the parting lines in the graphics area.

   Note the arrow direction. To specify a different draft direction for each segment of the parting line, click the name of the edge in the **Parting lines** box, and click **Other Face**.

NOTE: In this example, all parting lines were selected, using the same face.

Thin Feature

The thin features dialog box lets you define the characteristics of extruded thin features.

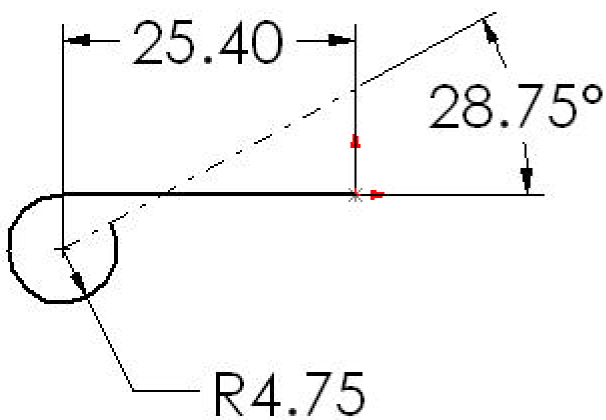
- **Type.** Type lets you specify whether to extrude the thin feature in **One Direction**, **Mid-plane**, or **Two Directions**.
- **Wall Thickness.** Use the spin box to specify the thickness of the thin feature wall.
- **Reverse.** Lets you reverse the extrusion of the wall thickness.
- **Link to Thickness.** Ensures that the boss is the same thickness as the base.

If you created a closed profile you can select the following options:

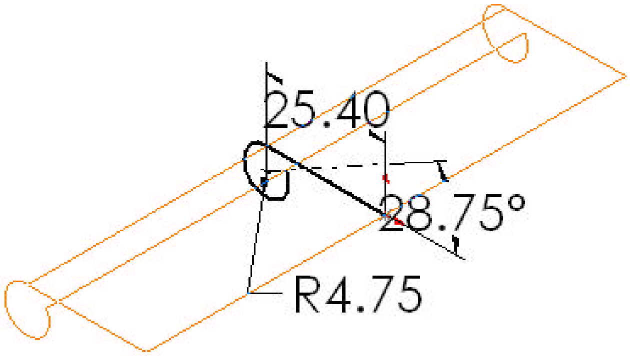
- **Cap Ends.** Specifies that the ends of the thin feature are capped. A capped thin feature must be extruded from a closed profile sketch. When the ends are capped, all walls of the feature are closed; the center is hollow.
- **Cap Thickness**. Lets you specify the thickness of the cap. (Available only if **Cap Ends** is selected.)

If you created an open profile you can select the following options:

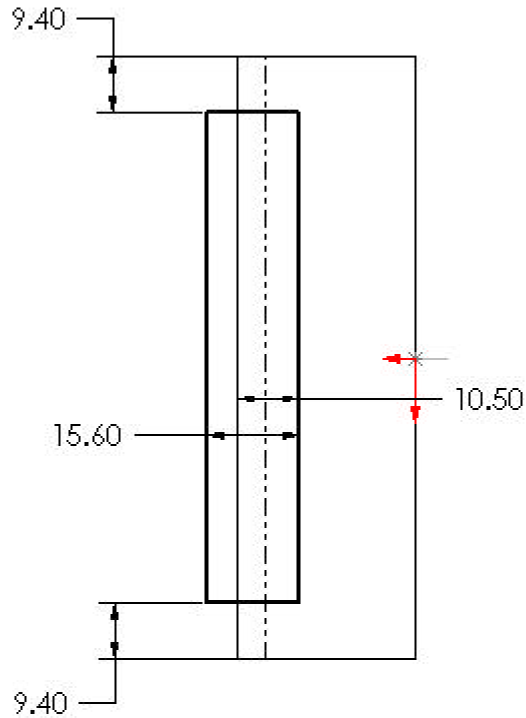
- **Auto Fillet.** Automatically creates a round at each edge where lines meet at an angle.
- **Fillet Radius.** Specifies the inside radius of the round.



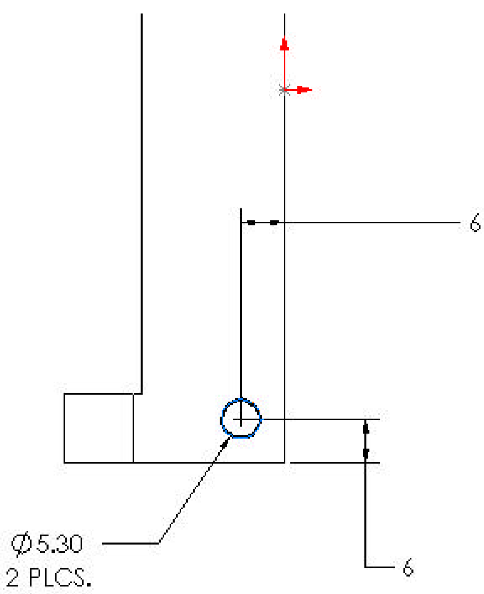
Thin Feature Base Extrude



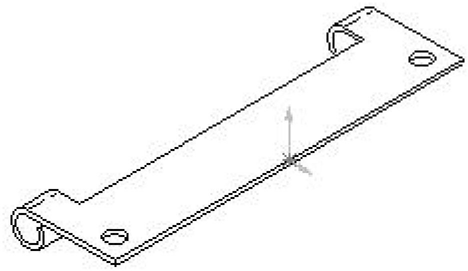
Extrude Midplane 102



Extrude Cut Feature



Extrude Cut Feature



Part finished with Mirror Patern Feature

Mirror Feature

Creates a copy of a feature (or multiple features), mirrored about a plane.

If you modify the original feature (**seed feature**), the mirrored copy is updated to reflect the changes.

To mirror a feature:

1 Click **Mirror Feature**  on the Features toolbar or **Insert**, **Pattern/Mirror**, **Mirror Feature**.

2 With the **Mirror plane** box selected, click on a plane or a face.

3 With the **Features to mirror** box selected, click one or more features in the model or in the FeatureManager design tree.

4 If you want to mirror only the geometry (faces and edges) of the features, rather than solving the whole feature, select **Geometry Pattern .**

NOTE: The geometry pattern option speeds up the creation and rebuilding of the pattern. However, you cannot create geometry patterns of features that have faces merged with the rest of the part.

5 Click **OK**.

Sheet Metal

Sheet metal parts are generally used as enclosures for components or to provide support to other components.

You can design a sheet metal part on its own without any references to the parts it will enclose, or you can design the part in the context of an assembly containing the enclosed components.

This chapter introduces the SolidWorks sheet metal functionality and describes:

- Designing sheet metal parts
- Rolling back and rebuilding sheet metal designs
- Creating sheet metal parts
- Adding and editing bends
- Selecting the type of bend allowance
- Adding and editing auto reliefs
- Adding features in the flattened state
- Creating rips
- Creating hems
- Creating bends across multiple tabs
- Creating flat pattern configurations
- Creating sheet metal drawings
- Creating forming tools

Bend Allowance Options

Bend Table

A sample bend table for sheet metal operations is provided in **<*install_dir*>\lang\english\sample.btl**. To use your own bend table, copy and edit this bend table using any text editor, to specify your required bend allowances.

sample.btl - Notepad

File Edit Search Help

```
Bend Allowance Tables
----------------------

Version: 1
Material: Steel
Units: meters


Thickness: 0.0005
Bend Radius (read across)      0.0000  0.0005  0.0010  0.0015  0.0020  0.0025  0.0030
Opening Angle (read down)
                            5  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           10  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           20  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           30  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           45  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           50  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           70  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           80  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                           90  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          100  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          110  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          120  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          130  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          140  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          145  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          160  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          170  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                          180  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002


Thickness: 0.0010
Bend Radius (read across)      0.0000  0.0005  0.0010  0.0015  0.0020  0.0025  0.0030
```

NOTE: Bend table units must be specified in meters. The sample bend table is provided only for informational purposes. The values in this table do not represent any actual bend allowance values. If the thickness of the part or bend angle falls between values in the table, the software interpolates the values to calculate the bend allowance.

K-factor

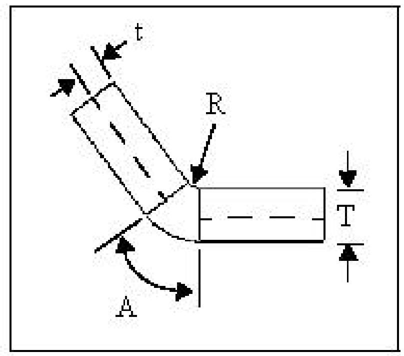
K-factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part.

Bend allowance using a K-factor is calculated as follows:

$BA=\Pi(R + KT)\ A/180$

where:

BA = bend allowance
R = inside bend radius
K = K factor, which is t / T
T = material thickness
t = distance from inside face to neutral sheet
A = bend angle in degrees (the angle through which the material is bent)



Bend Allowance Value

You can specify an explicit bend allowance for any sheet metal bend by entering the value when you create the bend.

Note: For a given bend radius and angle, the specified value for the bend allowance should be between the length of the inner edge and the length of the outer edge of the bend.

Sheet Metal Features

FeatureManager Design Tree

When you click **Insert Bends** or **Insert, Features, Sheet Metal, Bends**, two distinct stages are applied to the sheet metal part.

- The part is flattened and a bend allowance is added. The developed length is calculated, based on the bend radius and bend allowance.
- The flattened part is restored to the folded state to create the formed, folded version of the part.

Three new features and icons appear on the FeatureManager design tree that are specific to sheet metal operations. These features are:

- Sheet-Metal
- Flatten-Bends
- Process-Bends

These three features represent a process plan for the sheet metal part.

Sheet-Metal contains the definition of the sheet metal part. This feature stores the default bend parameter information (thickness, bend radius, bend allowance, auto relief ratio, and fixed entity) for the entire part.

Flatten-Bends represents the flattened part. This feature contains information related to the conversion of sharp and filleted corners into bends.

Each bend generated from the model is listed as a separate feature under **Flatten-Bends**. Bends generated from filleted corners, cylindrical faces, and conical faces are listed as **Round-Bends**; bends generated from sharp corners are listed as Sharp-Bends.

The **Sharp-Sketch** listed under **Flatten-Bends** is the sketch that contains the bend lines of all sharp and round bends generated by the system. This sketch cannot be edited but can be hidden or shown.

Process-Bends represents the transformation of the flattened part into the finished, formed part.

Bends created from bend lines specified in the flat are listed under this feature. **FlatSketch,** listed under **Process-Bends**, is a placeholder for these bend lines. This sketch can be edited, hidden, or shown.

Features that are listed after the **Process-Bends** icon in the FeatureManager design tree do not appear in the flattened view of the part.

You flatten the view of the part by rolling back the FeatureManager design tree to before the **Process-Bends** feature.

Rollback Bar

Reverts the model to an earlier state, suppressing recently added features. You can add new features or edit existing features while the model is in the rolled-back state.

You can roll back a part using the **Rollback Bar** in the FeatureManager design tree. The rollback bar is a wide yellow line which turns blue when selected. Drag the bar up or down the FeatureManager design tree to step forward or backward through the regeneration sequence.

You can use the **Rollback Bar** to flatten sheet metal bends, or you can click **Flattened**  to flatten. See also **Rolling Back and Rebuilding Sheet Metal Parts**.

To revert a part to an earlier state:

1 Place your cursor over the rollback bar in the FeatureManager design tree. The cursor changes to a hand.

2 Click to select the rollback bar. The bar changes color from yellow to blue.

3 Drag the rollback bar up the FeatureManager design tree until it is above the feature(s) you want rolled back,

- or -

Click in the FeatureManager tree and use the up and down arrow keys on the keyboard to move the rollback bar up or down.

TIP: To enable this use of the arrow keys, click **Tools**, **Options**, and on the **System Options** tab, select **FeatureManager**. Select the **Arrow key navigation** check box.

4 To roll forward again, drag the rollback bar to the bottom of the FeatureManager design tree.

NOTE: The FeatureManager icons are gray and unavailable after they are rolled back.

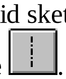
Edit Bends

You can edit bend information by using the **Edit Definition** option from the right-mouse menu. The scope of the changes depends upon which feature is edited. For more information, select from the following:

- Edit a single bend.
- Edit bends for an entire part.
- Edit bends listed under Flatten-Bends.
- Edit bends listed under Process-Bends.

NOTE: You cannot change the bend angle for sharp bends because this is calculated from your model.

You cannot change the bend angle and radius for round bends.
The bend angle is calculated from the model and the bend radius is specified in the sketch or in the fillet.

You must use a solid sketch **Line**  to add a bend to a flattened sheet metal part. You can not add a bend using a **Centerline** .

Edit a Single Bend

To edit a single bend:

1. In the FeatureManager design tree, right-click a **SharpBend**, **RoundBend**, or **FlatBend** item that you want to change, and select **Edit Definition**.
2. In the dialog box that comes up, make changes as needed to the bend **Radius**, **Angle**, or **Order**. Click **Bend Down** to reverse the direction of the bend.
3. In the **Bend Allowance** section, you can click to clear the **Use Default** checkbox if you want to change the use of the **bend table** , **k-factor**, or **bend allowance** .
4. To change the relief type and size, click the **Auto Relief** tab.
5. Click to clear **Use default relief**.
6. Change the type of relief cut in the **Relief Type** section.
7. If you select **Rectangular**, change the **Width** and **Depth** values in the **Relief Size** section.
8. Click **OK** to exit the dialog box and rebuild the part.

Edit Bends for Entire Part

To Edit bend parameters for the entire part:

1. In the FeatureManager design tree, right-click the **Sheet-Metal** feature, and select **Edit Definition**.
2. In the **Sheet-Metal** dialog box, change the **Default Bend Radius**.
3. Change the **Fixed edge or face** by selecting a different edge or face on the model.
4. In the **Bend allowance** section, change the use of the **bend table** , **k-factor**, or **bend allowance** .
5. To add auto reliefs, select **Use auto relief** in the **Auto Relief** section, and select the type of relief cut. To change the type of auto relief, select a different type of relief cut.
6. If you select **Rectangular**, you must specify a **Relief ratio**.
7. Click **OK**.

Edit Bends Listed Under Flatten-Bends

To edit bend parameters for all bends listed under Flatten-Bends:

1. In the FeatureManager design tree, select the **Flatten-Bends** feature.
2. Right click and select **Edit Definition**.
3. In the **Flatten-Bends** dialog box, clear the **Default bend radius.**
4. In the **Bend allowance** region, you can click to clear the **Use Default** checkbox if you want to change the use of the **bend table** , **k-factor**, or **bend allowance** .
5. Click **OK**.

Edit Bends Listed Under Process-Bends

To edit bend parameters for all bends listed under Process-Bends:

1. In the FeatureManager design tree, select the **Process-Bends** feature.
2. Right-mouse click and select **Edit Definition**.
3. In the **Process-Bends** dialog box, change the **Default Bend Radius.**
4. In the **Bend allowance** region, you can clear the **Use Default** checkbox if you want to change the use of the **bend table** , **k-factor**, or **bend allowance** .
5. Click **OK**.

Adding Walls to a Sheet Metal Part

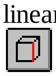
Important: All walls must be in place <u>before</u> the **Sheet-Metal** feature adds the bends.

It is recommended that you add all the walls as described here, before inserting bends. However, if necessary, you can rollback the model to the last feature before the **Sheet-Metal** feature to add a wall. Or, if parent-child dependencies allow, you may be able to re-order a wall before the **Sheet-Metal** feature.

NOTE: You cannot add walls to cylindrical or conical faces on sheet metal parts.

To add a wall to a sheet metal part:

1 Open a sketch on the face of the part where the new wall will be attached.

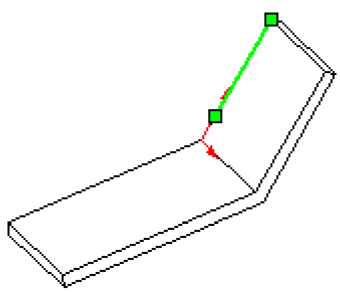
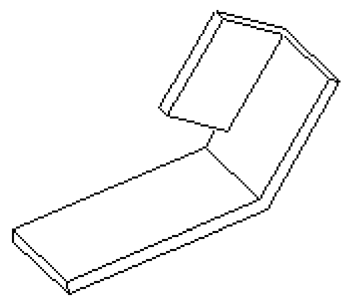
2 Select a linear edge of a planar face on the model to attach the wall to, and click **Convert Entities**  or **Tools**, **Sketch Tools**, **Convert Entities**.

3 Drag the vertices near existing bends a small distance away from the bends to allow for the bend radius.

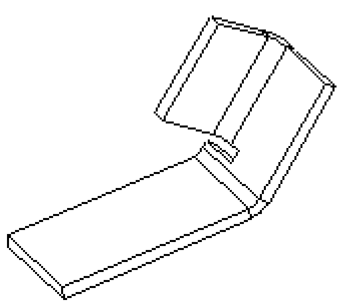
4 Click **Extruded Boss/Base**  or **Insert, Boss, Extrude**, then set the **Type** to **Blind** and specify the **Depth**.

5 On the **Thin Feature** tab, set the **Wall Thickness** to be the same as the base part.

6 Click **OK**.

7 Click  or **Insert, Features, Sheet Metal, Bends**.

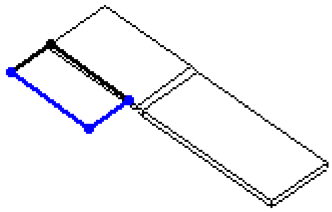


Extruding Tabs on a Sheet Metal Part

To extrude a tab on a flattened part:

1 Click a face on the flattened part and open a sketch.

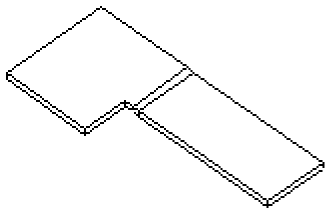
2 Sketch a profile with one side coincident with a model edge.



3 Click **Extruded Boss/Base**  or **Insert, Boss, Extrude**.

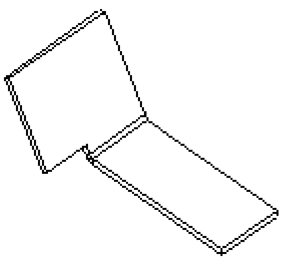
4 Click **Link to Thickness** to ensure that the boss is the same thickness as the base.

5 Observe the extrusion preview, and click **Reverse Direction**, if necessary.



6 Click **Flattened**  to rebuild the part.

The part rebuilds with the new feature in place.



Rip

Creates a rip feature along the selected model edges.

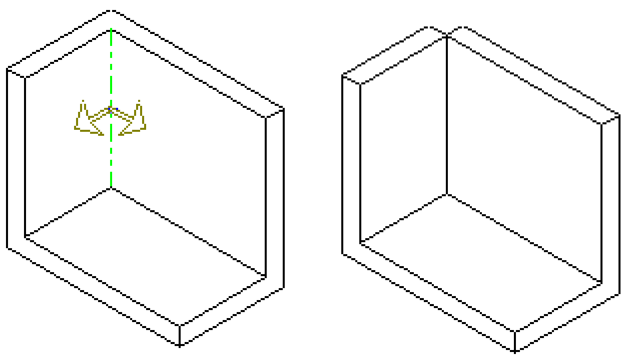
To create a rip feature:

1 On the model, select the inside linear edges to rip.

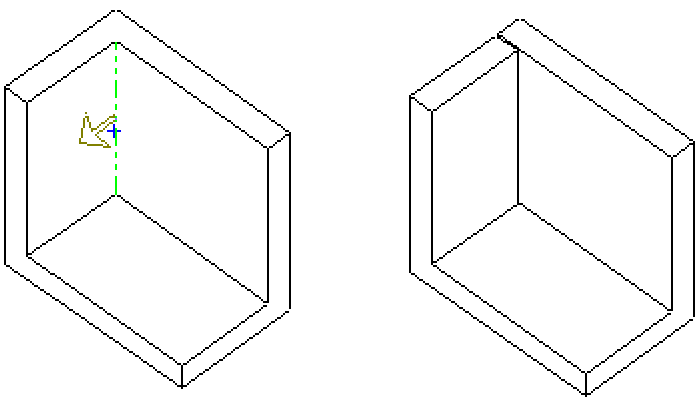
Note the arrows that appear on each selected edge. By default, rips are inserted in both directions. To insert a rip in only one direction, click the name of the edge listed under **Edge selection**, and click **Change direction**. Click **Change direction** again to insert the rip in the opposite direction.

2 Click **OK**.

Both directions:



One direction:



Auto Relief

The software automatically adds relief cuts wherever needed when inserting bends if you select **Use auto relief**. The software supports two types of relief cuts:

- Rectangular
- Tear

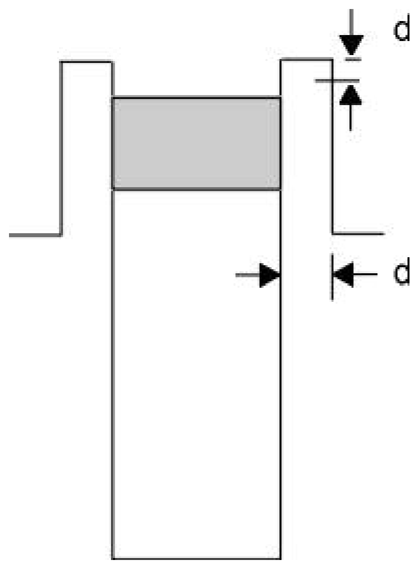
If you want to automatically add **Rectangular** reliefs, you must specify the **Relief ratio.**

Tear reliefs are of the minimum size required to insert the bend and flatten the part.

Relief Ratio

The distance **d** represents the width of the **Rectangular** relief cut and the depth by which the side of the **Rectangular** relief cut extends past a bend region, expressed as a ratio of material thickness.

The bend region is represented by the gray area of the diagram.



The value must be between 0.05 and 2.0. The higher the value, the larger the size of the relief cut added during insertion of bends.

Note:
You can edit a Rectangular relief cut to specify different width and depth dimensions.

Edit a Single Auto Relief

You can change the type and size of a relief cut for an individual bend.

NOTE: You cannot edit relief cuts in sheet metal parts created in versions prior to SolidWorks 99.

To change the type and size of a relief cut for a single bend:

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

Edit All Auto Reliefs

You can change the type of auto reliefs automatically added to the bend regions in sheet metal parts.

NOTE: You cannot edit relief cuts in sheet metal parts created in versions prior to SolidWorks 99.

To change the type of relief cut for all bends:

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

Creating Drawings of Sheet Metal Parts

You can create drawings of sheet metal parts using either one of the following methods:

- **Automatically** create the flat pattern configuration of the sheet metal part when you create the drawing.
- **Manually** create the flat pattern configuration of the sheet metal part, then create the drawing.

To show bend lines in flat pattern view:

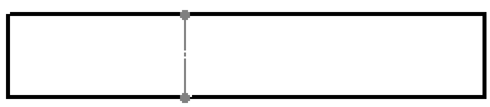
1 In the drawing window, expand **Drawing View** in the FeatureManager design tree to show the part's **Flatten-Bends** and **Process-Bends** features. Then expand both of these features.

2 Right-click **Sharp-Sketch** and click **Show,** then right-click **Flat-Sketch** and click **Show**.

To remove the display of the bend region lines:

1 In the drawing window, right-click **Drawing View** in the FeatureManager design tree, or right-click the drawing view in the graphics area.

2 Select **Tangent Edge, Tangent Edges Removed**.



To change the configuration in the drawing view:

1 Right-click the drawing view in which you want to change the configuration, and select **Properties**.

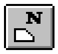
2 In the **Use named configuration** box, select a different configuration and click **OK**.

3 Right-click the drawing view again, and select **Update View**, if necessary.

Automatic Sheet Metal Drawings

To create a drawing of a flat pattern that the software creates automatically:

1 Create a new drawing document.

2 Click **Named View**  or **Insert, Drawing View, Named View**.

3 Change to the part window and click anywhere.

4 Select the Flat pattern view from the **Drawing View - Named View** dialog box, and click **OK**.

The software creates a flat pattern configuration automatically.

5 Return to the drawing window and click to place the flat pattern view.

The bend lines are automatically shown in the flat pattern view; the bend region lines are not shown.

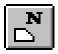
Multiple collinear edges in the flat pattern configuration are merged into a single linear edge in the drawing.

Manual Sheet Metal Drawings

To create a drawing of a flat pattern that you create manually:

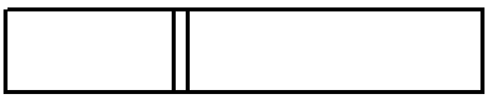
1 Create a **part configuration of a flat pattern**.

2 Create a new drawing.

3 Click **Named View**  or **Insert, Drawing View, Named View**.

4 Change to the part window and click anywhere. Then select the desired view from the **Drawing View - Named View** dialog box, and click **OK**.

5 Return to the drawing window and click to place the view on the sheet.



Notes

A note can be free floating or placed with a leader pointing to an item (face, edge, or vertex) in the document. It can contain simple text, symbols, parametric text, and hyperlinks.

To set **Note** options for the current document, click **Tools, Options, Document Properties, Notes**.

To create notes:

1. Click **Note** or **Insert, Annotations, Note**.
   The **Properties** dialog box for Note appears.
2. Type in text and select options as desired.
3. With the dialog box still open, click in the graphics area to place the note.
   - Click as many times as necessary to place multiple copies.
   - If the note has a leader, click once to place the leader, then click a second time to place the note.
   - You can change text and other items in the dialog box for each instance of the note.
   - While dragging the note and before placing it, press **Ctrl**. The note stops moving, but the leader continues, lengthening the leader. While still holding **Ctrl**, click to place the leader. Click as many times as necessary to place additional leaders. Release **Ctrl** and click to place the note.
4. Click **OK** to close the dialog box.

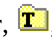
Once you have created a note and closed the dialog box, you can edit the note in place by double-clicking it. You can also edit the note and change its properties by right-clicking the note and selecting **Properties**. For more information on the options available in the **Properties** dialog box..

You can add more leaders to an existing note by holding down **Ctrl** and dragging a leader attachment point.

Annotation Properties

Lets you select the types of annotations that you want to display and set text scale and other annotations options.

To select the types of annotations to display:

1. In the FeatureManager design tree, right-click the **Annotations** folder, , and select **Details**.
2. In the **Annotation Properties** dialog box, specify a **Display Filter** by selecting the annotation types to display by default:

Cosmetic threads	**Geometric tolerances**
Datums	**Notes**
Datum targets	**Surface finish**
Feature dimensions	**Welds**
Reference dimensions	

   \- or -

   Select **Display all types** to show all kinds of annotations available for that part or view.
3. Change the values in the **Text scale** edit box to change the scale of the text used in the annotations.
4. Select from the following options:
   - **Always display text at the same size.** When checked, all annotations and dimensions are displayed at the same size regardless of zoom. Note that drawings have this option disabled and always zoom the text height.
   - **Display items only in the view orientation in which they are created.** When checked, any annotation is displayed only when the model is viewed in the same orientation as when the annotation was added. Rotating the part or selecting a different view orientation removes the annotation from the display.
   - **Display annotations / Display assembly annotations.** When checked, all annotation types that are selected in the **Display filter** are displayed. For assemblies, this includes not only the annotations that belong to the assembly, but also the annotations that are displayed in the individual part documents.
   - **Use assembly's setting for all components.** When checked, the display of all annotations matches the setting for the assembly document, regardless of the setting in the individual part documents. Use this option along with **Display assembly annotations** to display different combinations of annotations.
   - If you use JIS dimensioning standards, specify the **JIS surface finish size** (1,2,3 characters) or **Scale**.
5. Click **OK** to accept your changes; click **Cancel** to exit the dialog without saving the changes.

ANSI Weld Symbol

Creates symbols to provide weld specifications on your part, assembly, and drawing documents.

To create a weld symbol:

1. Click an edge in the drawing view where you want to indicate a welded joint.
2. Click **Weld Symbol** or **Insert, Annotations, Weld Symbol**.
   The ANSI Weld Symbol dialog box appears.
   The symbol is displayed in the preview box as you select symbols and add appropriate dimensions and values.
3. In the boxes beside the **Weld Symbol** button, enter the weld dimensions.
4. Click the **Weld Symbol** button and select a symbol type from the list of standard symbols.
5. New! For a second fillet, select the **2nd fillet** check box and enter the weld dimensions in the boxes beside the check box.
   The **2nd fillet** is available for only certain weld symbols (**Square** or **Bevel**, for example).
6. Specify other options as needed, such as **Finishing Method, Contour, Joint with Spacer**, **Groove angle**, or **Root opening**. For **Root opening**, you can select the **Inside** check box, the **Arrow** check box, or neither.
7. Select check boxes for **All around**, **Field or site weld**, **Display pointing down**, or **Stagger weld** as needed.
8. To specify the welding process, click **Specification process** and enter the text in the box.
9. Choose whether the weld is to be applied to the object on the **Arrow Side** or the **Other side**.
10. Choose a **Leader Anchor** style.
11. New! If you are in a drawing with more than one layer, you can choose a named layer from the **Layer** list.
12. Click **OK**.

Geometric Tolerancing

The SolidWorks software supports the ANSI Y14.5 Geometric and True Position Tolerancing guidelines.

- You can place geometric tolerancing symbols, with or without leaders, anywhere in a drawing, part, assembly, or sketch, and you can attach it anywhere on a dimension line.
- The pointer changes to when it is near the handle of an arrow of a geometric tolerancing symbol on a dimension.
- You can add multiple symbols without closing the dialog box, and you can display multiple perpendicular leaders.

To create geometric tolerancing symbols:

1 Click **Geometric Tolerance** , or click **Insert, Annotations, Geometric Tolerance**.

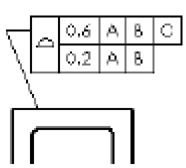
2 Type in values and select symbols. As you add items, a preview is displayed.

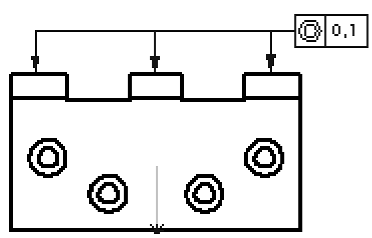
For more information on the options available (Geometric Characteristic Symbols, Material Condition Symbols, Projected tolerance zone, and so on). For information on leader, arrow, anchor, and font options, see **Geometric Tolerance Details**.

3 Click to place the symbol.

- Click as many times as necessary to place multiple copies.
- If the symbol has a leader, click once to place the leader, then click a second time to place the symbol.
- You can change text and other items in the dialog box for each instance of the symbol.
- While dragging the symbol and before placing it, press **Ctrl**. The note stops moving, but the leader continues, lengthening the leader. While still holding **Ctrl**, click to place the leader. Click as many times as necessary to place additional leaders. Release **Ctrl** and click to place the symbol.

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





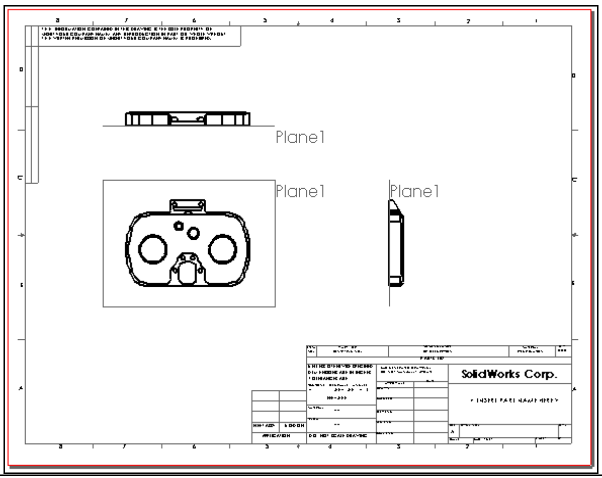
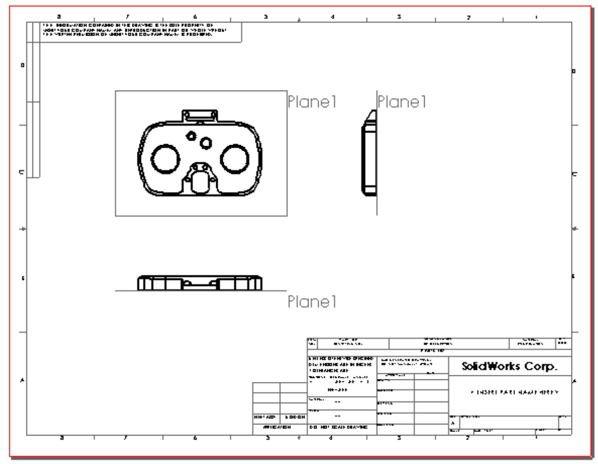
You can drag the symbol any place in the document.

To edit an existing symbol, double-click the symbol, or right-click the symbol and select **Properties**.

You can add more leaders to an existing note by holding down **Ctrl** and dragging a leader attachment point.

First Angle and Third Angle Projection

In third angle projection, the default front view from the part or assembly is displayed at the lower left. In first angle projection, the front view is displayed at the upper left. For information about setting the default projection:

	
3rd Angle Projection	**1st Angle Projection**

Sheet Setup

Lets you change the setup of the active drawing sheet.

To change the setup of a drawing sheet:

1 While in an active drawing document, click **Edit**, **Properties**, or right-click the sheet, and select **Properties**.

2 You can make the following changes in the **Sheet Setup** dialog box:

- Enter a title in the **Name** box.
- Select a **Paper Size** from the list of standard paper sizes or select **UserDefined** to specify a custom paper size.
- Displays the **Height** of the selected standard paper size, or lets you specify the custom paper size **Height**.
- Displays the **Width** of the selected standard paper size, or lets you specify the custom paper size **Width**.
- Specify the **Scale**. The default is 1:2. (The scale is displayed in the status line at the bottom of the drawing.)
- Select a standard sheet format, or select **Custom** or **None**.
- Use the browse button to locate and use a custom drawing sheet format.
- **Reload Sheet Format**. If you make changes to the sheet format, you can return to the default format by clicking this button.
- Select the **Type of Projection** that will be used if you use the **Standard 3 View** option on this sheet.
  - **First Angle.** Orthographic views in first angle projection.
  - **Third Angle.** Orthographic views in third angle projection (the default).
- Specify the letter of the alphabet that will be used for the next section or detail view in the **Next View Label** box.
- Specify the letter of the alphabet that will be used for the next datum symbol in the **Next Datum Label** box.
- Under **Use custom property values from model shown in**, select a view from the list. The properties defined for the model shown in the selected view are used to populate any notes that are linked to properties. See **Linking Notes to Document Properties**.

3 Click **OK**.

Modifying the Sheet Setup

You can set up the sheet details when you start a drawing, or later. You can also modify existing sheet details. If you choose **No Sheet Format** when you open a new drawing, the default values specified in the **Drawings Options** are used.

To specify sheet details:

1 Right-click the sheet icon in the FeatureManager design tree, or any blank area of the drawing sheet, or the sheet tab at the bottom of the drawing window, and select **Properties**.

You can edit the properties only of the active sheet. For information about creating and activating additional sheets, see **Multiple Drawing Sheets**.

2 Enter your choices in the **Sheet Setup** dialog box:

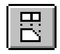
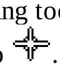
- **Name**. Enter a title in the box.
- **Paper size**. Select a standard size from the list, or select **User Defined** to specify a custom paper size. If you selected **User Defined**, specify a **Height** and **Width** for the paper.
- **Scale**. Specify the default scale for all views on the sheet.
- **Sheet Format**. Select a standard sheet format from the list, or select **Custom** or **None**. If you select **Custom**, use the **Browse** button to select a custom sheet format.
- **Reload Sheet Format**. If you make changes to the Sheet Format, you can return to the default format by clicking this button.
- **Type of projection**. Select **First angle** or the default, **Third angle**.
- **Next view label, Next datum label.** Specify the letter of the alphabet to be used.
- **Use custom property values from model shown in**. This option is used only if more than one model document is shown on the sheet. If the drawing has notes that are linked to custom file properties of a model, select which view contains the model whose properties you want to use. If you do not specify otherwise, the properties of the model in the *first* view added to the sheet are used. See **Linking Notes to Document Properties**.

3 Click **OK**.

Projection View

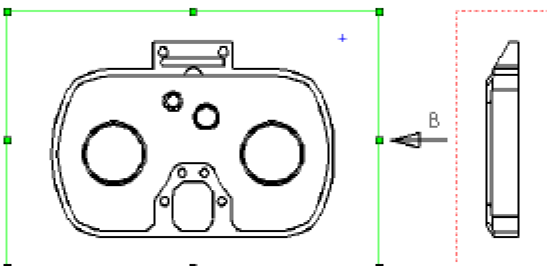
A **Projection View** is created by projecting an orthographic view, using **First angle or Third angle projection**, as specified in the **Sheet Setup** dialog box.

Creating a projection view:

1. Select an existing view by clicking inside its boundary.
2. Click **Projected View**  on the Drawing toolbar, or click **Insert**, **Drawing View**, **Projection.** The pointer shape changes to .
3. To select the direction of projection, move the pointer to the appropriate side of the selected view.

   A preview of the view is displayed, snapped to the nearest projection. To override the snapping behavior, hold the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

   You can project to the left, right, above, or below (in this example, the user moved the pointer to the right of the selected view).

4. When the view is where you want it to be, click to place the view.

   The projection view is placed on the sheet, aligned to the view from which it was created. By default, you can move a projection view only in the direction of the projection. For information about changing the alignment of views, see **View Alignment and Display**.
5. To display an arrow indicating the direction of projection, right-click the projection view on the sheet or in the FeatureManager design tree, and select **Properties**. Click the **Display view arrow** check box and enter a label if desired (maximum of two characters).

Projection Views are linked from the child view to the parent. Right-click the Projection View and select **Jump to Parent View**.

Manintaing Company Standards

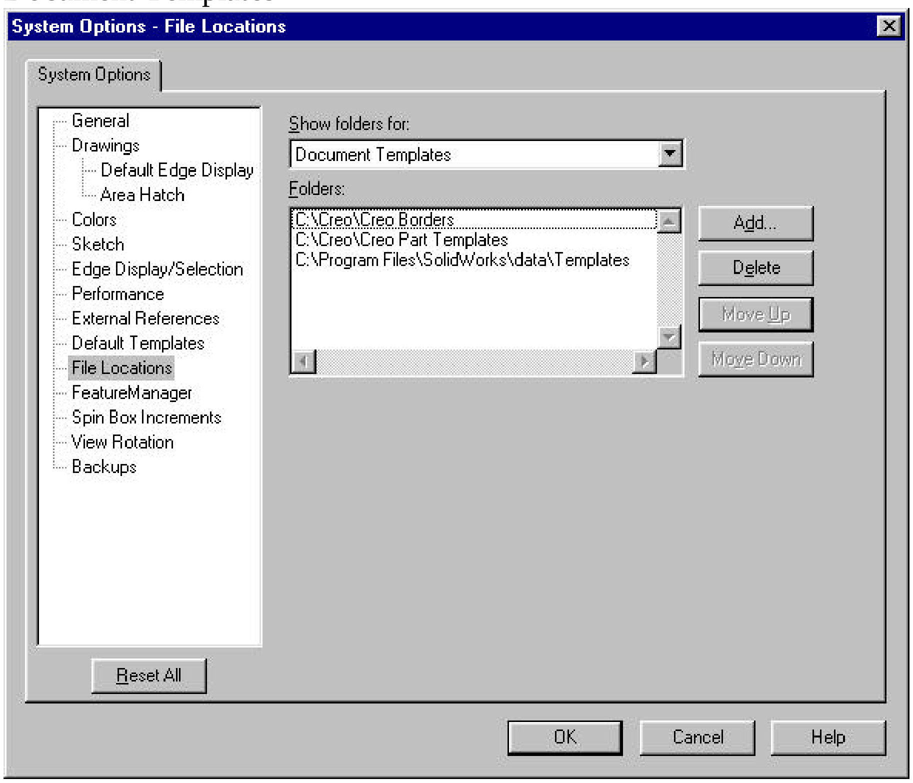
Use of Template files

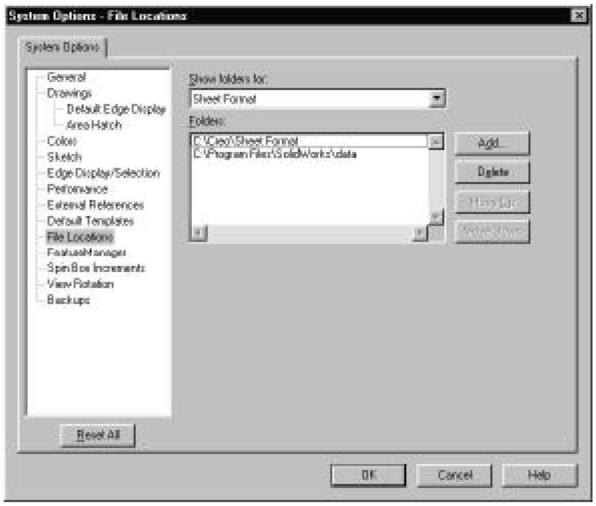
This exercise focuses on how to create template files from existing drawing files. Basiclay, the technique involves changing exiting drawing file extensions from .SLDDRW to .DRWdot. Next, the file path location is added to the Document Templates listing in File locations (shown below).

1 In the directory c:\Creo\Creo Borders, Rename: Creo-B.SLDDRW to: Creo-B.DRWdot
2 From the Pulldown menus select Tools | Options
3 Select the System Options Tab
4 Select File Locations
5 From the Show folders for: list select Document Templates
6 Press the Add Button
7 Find the directory C:\Creo\Creo Borders

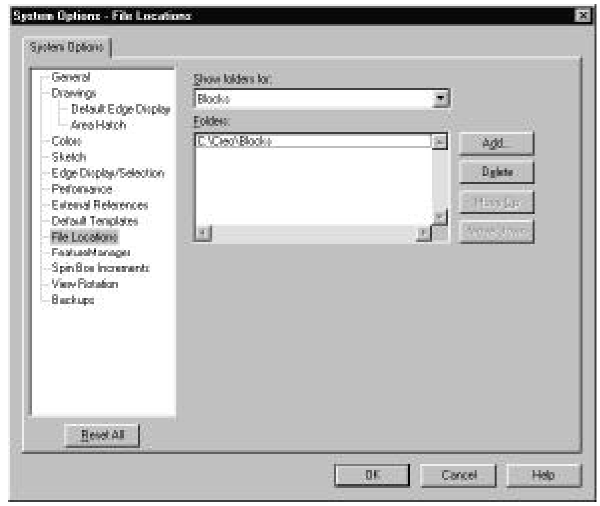
Standard Borders

Document Templates





Standard Sheet Format



Standard Blocks

Linking Notes to Document Properties

To automatically insert information in a drawing, you can link note text in the drawing sheet to document properties.

All SolidWorks documents have the system-defined property **SW-File Name** (the name of the document, without the extension). Additionally, drawings have the following system-defined properties:

Property Name	**Value**
SW-Current Sheet	sheet number of the active sheet
SW-Sheet Format Size	sheet size of the active sheet format
SW-Sheet Name	name of the active sheet
SW-Sheet Scale	scale of the active sheet
SW-Total Sheets	total number of sheets in the active drawing document

You can link a note to the properties of the model that is shown in the drawing (the **SW-File Name** property, or a user-defined custom property in the model document).

While you are editing the sheet format, a variable for the property name is displayed (**$PRP: "property name"**). When you return to editing the sheet, the value of the property is displayed, if it is found. If the property value cannot be found, the variable for the property name is displayed on the sheet.

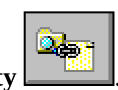
A linked note can include additional text, and it can include links to more than one property. For example, to display the current sheet number and the total number of sheets, you can add this note:

SHEET $PRP: "SW-Current Sheet" OF $PRP: "SW-Total Sheets"

On the sheet, the property values are displayed:

SHEET 1 OF 2 (on the first sheet of a two-sheet drawing)

To link a note in the sheet format to a property:

1 In the **Properties** dialog box for the **Note**, click **Link to Property** .

2 In the **Link to Property** dialog box, select a **Property Name** from the list. The list contains the names of the system-defined properties, and any custom properties to which you have assigned values in the active drawing document.

3 To link the note to the properties of the model shown in the drawing, select the **External model reference** check box, then select a **Property Name** from the list. The value of the property for the model shown in the first view inserted into the sheet is displayed by default.

If the sheet contains views of more than one model, you can specify which view contains the model whose properties you want to use. Right-click the sheet, select **Properties**, then select the view from the list under **Use custom property values from model shown in**.

If no model is shown in the drawing (for example, when you are creating a new sheet format), you can either select a property from the drawing's list, or add a new custom property. The property is resolved when a model is added to the sheet.

When you save the sheet format, any new properties that you added are saved with the sheet format. The properties are added to any new drawing that uses the sheet format.

4 **Update** the drawing, if necessary.

Cosmetic Threads

You can represent threads on a part, assembly, or drawing, and you can attach a thread callout note. You can add cosmetic threads to conical holes. If the conical thread does not end at a flat face, it is trimmed by the curved face.

A cosmetic thread differs from other annotations, in that it is an absorbed feature of the item to which it is attached. For example, the cosmetic thread on a hole is in the FeatureManager design tree under the **Hole** feature, along with the sketches used to create the hole.

Cosmetic threads added in a part or assembly can be imported to a drawing view. If you add a cosmetic thread while working in a drawing view, the part or assembly is updated to include a **Cosmetic Thread** feature.

For tap and pipe tap holes, you can add cosmetic threads in the **Hole Wizard**.

To insert cosmetic threads:

1. On a cylindrical feature (a boss, a cut, or a hole), click the circular edge where the thread begins. If the hole is conical, select the major diameter.
2. Click **Cosmetic Thread** on the Annotations toolbar, or click **Insert, Annotations, Cosmetic Thread**.
3. In the **Cosmetic Thread Properties** dialog box, select the thread to apply.
4. Click **OK**.

To edit a cosmetic thread:

1. In a part or assembly document, right-click the **Cosmetic Thread** feature, and select **Edit Definition**.

   You can select the feature either in the graphics area, or in the FeatureManager design tree.
2. Make the necessary changes in the **Cosmetic Thread** dialog box, and click **OK**.

To edit the thread callout on a cosmetic thread:

If you only need to edit the text of the thread callout note, double-click the note to edit in place. You can also right-click the note and select **Properties**.

To specify the line style and weight for cosmetic threads in the active drawing document:

1. Click **Tools, Options**. On the **Document Properties** tab, select **Line Font**.
2. In the **Type of edge** section, select **Cosmetic Thread**.
3. Choose a **Style** and **Thickness** from the lists.

The **Preview** box shows you the results

Stacked Balloons

Stacked balloons have only one leader for a set. You can stack the balloons vertically or horizontally and up or down, or left or right.

You can insert a stacked balloon without selecting a component, so you can annotate an item that is part of an assembly but not actually modeled, such as glue or a liquid. A question mark appears in the balloon. Edit the question mark to replace it with your text.

If you change the **Item Number** in a stacked balloon, the item number in the bill of materials also changes.

You can also insert balloons that are not stacked.

To insert stacked balloons:

1. Click **Stacked Balloon** on the Annotation toolbar or click **Insert, Annotations, Stacked Balloon**.
2. Select the part where you want the balloon leader.
3. Continue to select parts.

   A balloon is added to the stack for each part selected. As you are adding the stacked balloons, you can right-click any balloon in the stack, select **Stack Direction**, and choose a new stack direction (**Up**, **Down**, **Left**, **Right**).

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

To add balloons to the stack later:

Right-click one of the stacked balloons and select **Add to Stack**.

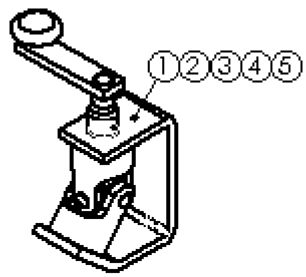
To edit balloon text:

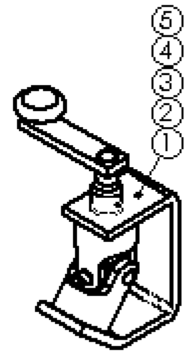
Double-click the balloon text and edit in place.

To change the properties of the stacked balloons:

1. Right-click any balloon in the stack and select **Properties**.

   The **Note Properties** dialog box appears.
2. Change the following properties:
   - In the **Balloon note text** section, select the type of text from the **Upper** or **Lower** lists, or both. (**Lower** is available only if you select the **Circular Split Line** style.)
   - In the **Stack Data** section, select **Up**, **Left**, **Down**, or **Right**. Set the maximum stack length in **Balloons per line**.
   - In the **Leader** section, choose **Always show leaders** or **No leaders**. Select or clear the **Display with bent leader** check box.
   - Clear the **Smart** check box to choose an **Arrow style** from the list and choose **Closest**, **Left**, or **Right** for the **Leader anchor**.
   - In the **Balloon** section, select the **Style** and **Size** from the lists. The **Style** of **None** is not available for stacked balloons.
3. Click **OK**.


1 2 3 4 5


5
4
3
2
1

Bill of Materials

Lets you insert a bill of materials into the drawing of an assembly. If you add or delete components in the assembly, the bill of materials automatically updates to reflect the changes if you select the **Automatic update of BOM** option under **Tools, Options, System Options, Drawings**.

NOTE: To insert a bill of materials in a drawing, you must have Microsoft Excel 97 or later installed on your computer.

To insert a bill of materials:

1 With a drawing view selected, click **Insert, Bill of Materials**.

2 In the **Select a BOM Template** dialog box, select the Excel template for the BOM.

The default BOM template is in ***install_folder*\lang\english\Bomtemp.xls**. You can customize the template by adding columns or **changing the text formatting**. You can also create new templates to meet your company's standards.

3 Select from the following options:

- **Use summary info title as part number**. If you assigned a part identifier number in the title box of the **Summary Info** for the part, you can use that identifier in the bill of materials.
- **Use the document's note font when creating the table.** When selected, the BOM uses the text font specified for notes, under **Tools, Options, Document Properties, Detailing, Notes, Font**. Otherwise, the font specified in the template is used.

4 Specify the way to list sub-assemblies and their components in the bill of materials:

- **Show parts only**. Select this option to list only parts in the bill of materials. Subassemblies are not listed; their components are listed as individual items.
- **Show top level subassemblies and parts only**. Select this option to list parts and subassemblies in the bill of materials. Each subassembly type is an item; the individual subassembly components are not listed.
- **Show subassemblies and parts in an indented list**. Subassemblies are listed as items; subassembly components are shown below the subassembly to which they belong, but without item numbers.

5 Under **Anchor point**, select from the following options:

- **Use table anchor point**. The standard templates supplied with the software include an anchor point for placing the BOM in a specific location on the sheet. To use the point, select this option, then select which corner of the BOM table to make coincident with the point (top right, bottom left, etc.). If you do not use the anchor point, the BOM is placed on the sheet near the selected view. See **Anchor Point for Bill of Materials** .
- **Add new items by extending top border of table**. New components are always added at the bottom of the table, and the rest of the table shifts upward. The last row of the table remains in the same position on the drawing sheet. This option applies only when you do not use an anchor point.

6 Click **OK**.

A bill of materials is displayed that lists the components of the assembly.

To move the bill of materials on the drawing sheet, drag it to where you want to place it.

Bill of Materials - Custom Properties

You can add more information such as Material, Vendor_Number, Cost, and so on to the open part or sub-assembly document by clicking **File**, **Properties**, and selecting the **Custom** tab or the **Configurations specific** tab. (See **Summary Info - Custom** for information about specifying properties.)

Then, you can add columns for these properties to the default bill of materials template (found in ***install_directory*\lang\english\Bomtemp.xls)**, or to a custom template that you create. When the bill of materials is inserted into an assembly drawing, it will be automatically populated with the values for the properties you defined on the property sheet.

To add custom columns to the bill of materials:

1 Open the BOM template file. To add the columns to the default BOM template, open ***install_directory*\lang\english\Bomtemp.xls**.

To create a new BOM template, it is a good idea to make a copy of the default template, then open the copy.

2 Insert a new column where you want the information to appear. The new column must be to the left of the column marked **$$END**.

3 Enter the column heading for the property you want to display in that column, and press **Enter**. The name appears in the **Formula Bar** in the Excel toolbar. This column heading does not need to match the custom property name.

4 Click the cell to select it, then click **Insert, Name, Define** and type in the name of the property that you want to appear in this column. This cell name replaces the default alphanumeric cell name that appears in the **Name Box** in the Excel toolbar whenever you click a cell.

The cell name is listed in the **Names in workbook** list for the selected cell. This name must match the name of the property from the part or sub-assembly document, and must not contain any spaces.

5 Click **Add**, then click **OK**.

When you add a bill of materials to a drawing, it will now include the added column. The cells will be populated with the values of the custom properties of the document, or the values of the configuration specific properties for the item.

NOTE: Do not change the cell names in the **Name Box** for the default columns in the BOM template. You may change the text in the column heading, but not the cell name.

Summary Info – Custom

Summary Info – Configuration Specific

Allows you to specify custom properties for the part, assembly, or drawing document, or configuration specific properties for the active part or assembly configuration.

You can use the properties in several ways:

- In a **bill of materials**
- As advanced selection criteria for **managing assembly configurations**
- In drawing **sheet formats** and to **link notes to document properties**

To enter custom properties:

1. Enter a **Name** for the property, or choose one from the list. If you plan to use the property as a custom column in a Bill of Materials, do not include any spaces in the **Name**.
2. Select the **Type** of value the custom property will use.
3. Enter a **Value** for the custom property that is compatible with the selection in the **Type** box.
4. Click **Add**.

   The **Properties** box displays the name, value and type of the custom properties.

To modify custom properties:

1. In the **Name** box, click the property in the **Name** box that you want to change.
2. Edit the **Type** or **Value** as needed.
3. Click **Modify**.

To delete custom properties:

1. Click the property in the **Properties** box that you want to delete.
2. Click **Delete**.

Detailing Overview

You can add much of the necessary detailing of your models in part and assembly documents. This includes dimensions, notes, symbols, and so on. Then you can insert the dimensions and annotations from the model into a drawing.
Once in the drawing, you can add reference dimensions, other annotations, and a bill of materials if required. Annotations and reference dimensions added in a drawing do not affect the part or assembly document.

Insert Model Items

Inserts dimensions, annotations, and reference geometry from the model into the current drawing.

You can insert items into a selected feature, an assembly component, a drawing view, or all views. When inserting into all views, dimensions and annotations appear in the most appropriate view. Features that appear in partial views, such as **Detail** or **Section** views, are dimensioned in those views first. Inserting annotations this way reduces the effort required to clean up the drawing.

To insert existing annotations or reference geometry into a drawing:

1 To annotate a single feature or assembly component, select the item.

To annotate a single drawing view, select the view.

To annotate all the views at once, do not select anything.

2 Click **Model Items** on the Annotation toolbar, or click **Insert, Model Items**.

3 From the **Insert Model Items** dialog box, select the types of items to insert:

- **Annotations -** Cosmetic Threads, Datums, Datum Targets, Dimensions, Geometric Tolerances, Notes, Surface Finish Symbols, Weld Symbols. If you select **Dimensions**, you have the option of also inserting **Instance/Revolution Counts** if those dimensions exist in the part or assembly.
- **Reference Geometry -** Axes, Curves, Planes, Surfaces, Piping Points, Origins.

- or -

4 Click the **All Types** check box to insert all of the annotations and reference geometry.

5 New! Select a layer from the **Layers** list (available if your drawing has layers) for the items to be inserted only in the specified layer.

6 From the **Import From** box, select one of the following options:

- **Entire Model.** Displays all of the selected item types that exist in the model.
- **Selected Component.** If this drawing is of an assembly, displays the items that exist on the selected component only.
- **Selected Feature.** Displays the annotations existing on the selected feature only.

7 Change the selection of views if necessary. To insert dimensions into selected views, clear the **Import Items into All Views** check box, then click the desired views in the graphics area or FeatureManager design tree.

The selected views are listed in the **Import into Views** box. Click the views again to remove them from the list; right-click the drawing sheet, and select **Clear selections** to remove all the views from the list.

NOTE: When **Selected Feature** or **Selected Component** is checked, annotations for the item are displayed only in the view where you selected the item. You cannot select a different view or all views.

8 To prevent the insertion of annotations that belong to hidden model items, clear the **Include dimensions from hidden features** check box. Annotations on features that are completely hidden by other geometry are not inserted. The insertion operation is slower, but the resulting views do not contain annotations that you may not want.

9 Click **OK** to insert the items; click **Cancel** to exit without making the changes.

Attachment points of imported annotations can be dragged, but not re-attached to another edge, face, vertex, and so on.

You can toggle the visibility of individual reference geometry items. Right-click the item, and select **Hide** or **Show**.

NOTE: Imported annotations display in the default **Imported Annotation** color; reference annotations (added in the drawing) are displayed in the default **Reference Annotation** color. These colors are specified in **Tools**, **Options**, **System Options**, on the **Color** page.

Moving and Copying Dimensions

Once dimensions are displayed, you can move them within a view or to another view. When you drag a dimension from one place to another, the dimension reattaches to the model, as appropriate. You can only move or copy dimensions to a view where the orientation is appropriate for that dimension.

- To move a dimension within the view, drag the dimension to the new position.
- To move a dimension from one view to another view, hold the **Shift** key as you drag the dimension into another view.
- To copy a dimension from one view to another view, hold the **Ctrl** key as you drag the dimension into another view.
- To move or copy several dimensions at once, hold the **Ctrl** key as you select

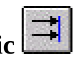
Aligning Dimensions

To select a group of dimensions:
You can select a group of dimensions in a drawing view either by holding the left mouse button and dragging a box around the dimensions or by holding Ctrl while selecting them. The selected dimensions must be of the same type.

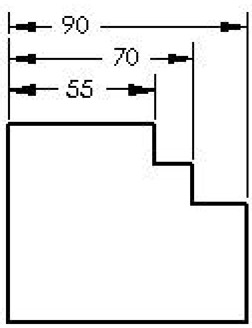
Align Dimensions Parallel/Concentric

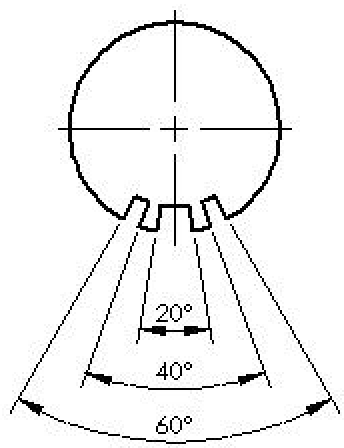
In a drawing view, aligns and groups selected linear, radial, or angular dimensions, with uniform spacing. The selected dimensions must be of the same type.

To align and group parallel dimensions:

1. In a drawing view, hold the **Ctrl** key and click two or more dimensions that you want to align. You can also select the group of dimensions by holding the left mouse button and dragging a box around the dimensions.
2. Click **Align Parallel/Concentric**  or **Tools, Dimensions, Align Parallel/Concentric**.

The dimensions are arranged with a uniform distance between the arrows. They also are grouped, and retain the parallel spacing when moved.





Aligned Parallel **Aligned Concentric**

To specify the distance between dimensions:

1. Click **Tools**, **Options**. On the **Document Properties** tab, under **Detailing**, click **Dimensions**.
2. Under **Offset distances**, specify a value for **From last dimension** (the distance between dimensions).

The **From model** value (the distance between the first dimension and the model) is not used with **Align Dimensions Parallel**; it is used only with **baseline dimensions**.

Center Marks

You can place axis lines for showing center marks on circles or arcs that can be used as reference points for dimensioning. You can set options for the default size of center marks and choose whether or not to display extended axis lines. Click **Tools, Options**. On the **Document Properties** tab, select **Detailing**. In the **Center marks** section, type a size and select or clear **Show lines**. The options apply to the active document.

Center marks are available in drawings only.

Center marks in Auxiliary Views are oriented to the viewing direction such that one of the lines of the Center Mark is parallel to the view arrow direction.

To create center marks:.

1. Click **Center Mark** on the Annotations toolbar, or **Insert, Annotations, Center Mark**.
2. Click the model edges (circles or arcs) or silhouettes.

To change the display attributes of individual center marks, right-click the center mark and select **Properties**.

Hole Callouts

Hole callouts are available in drawings. If you change a hole dimension in the model, the symbol updates automatically.

To add a hole callout symbol:

1. Click **Hole Callout** on the Annotations toolbar, or click **Insert, Annotations, Hole Callout**.
2. Click the edge of a hole.

   The **Modify Text of Dimension** dialog box appears, with symbols appropriate to the selected hole.

   You can add text (for example, the number of places where the hole occurs). However, do not change the dimensions or symbols for the size and type of hole in this dialog box.
3. Modify as necessary, and click **OK.**

When you right-click the hole callout and select Properties, the Dimension Properties dialog box appears. You can change properties of the hole callout in the dialog box. Click Modify Text to bring up the Modify Text of Dimension dialog box.

RapidDraft Drawings

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

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

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

When a RapidDraft drawing is out of sync with its model, it prints with a water mark that states:

SolidWorks RapidDraft – Out-of-Sync Print

Advantages of RapidDraft Drawings

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

Performance

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

File Size

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

Updating Views

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

SolidWorks Explorer -- Overview

The SolidWorks Explorer is a file management tool designed to help you perform such tasks as renaming, replacing, and copying SolidWorks files. You can show a document's references, search for documents using a variety of criteria, and list all the places where a document is used. Renamed files are still available to those documents that reference them.

NOTE: The SolidWorks Explorer is not a PDM (Product Data Management) tool; however, it does perform many useful tasks and simplifies many file management processes.

You can use SolidWorks Explorer with or without the SolidWorks application.

To activate the SolidWorks Explorer:

- From within the SolidWorks application, click **Tools**, **SolidWorks Explorer**.
- From the Windows Start menu, click **Programs**, **SolidWorks 2000**, **SolidWorks Explorer**.
- Create a shortcut by dragging the **SolidWorks Explorer** icon to your desktop.

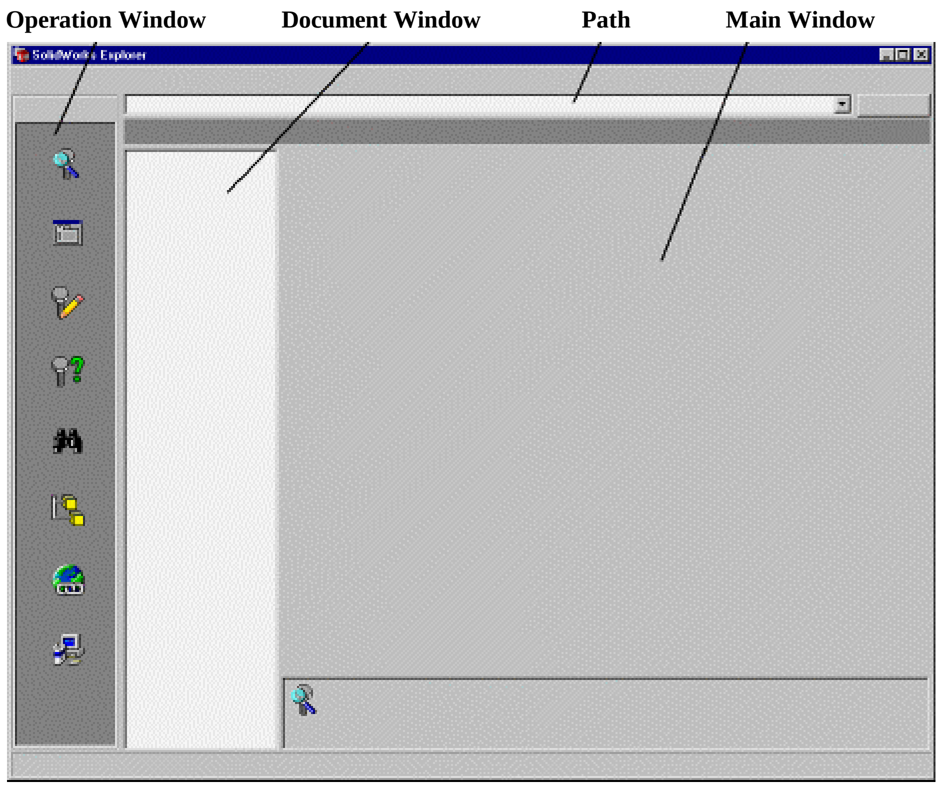
The SolidWorks Explorer window has several main features:

- **Path**. At the top of the SolidWorks Explorer window is a box that displays the drive letter, path, and name of the active document. Initially this is empty.
  - Click **Browse** to locate a document.

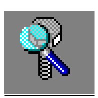
  \- or -

  - Drag a document from the Windows Explorer to the **Document** window.
- **Document** window. This window lists the active document and its parent or child documents. Click on these document icons to select the documents that you want to manage.

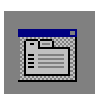
  To change to another document, drag a different document from Windows Explorer or browse to a different document in the **Path**.
- **Main** window. This window displays the dialogs or graphics that are appropriate for the operation that you select.
- **Operation** window. This window displays the icons that represent the file management tasks that you can perform with SolidWorks Explorer.



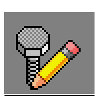
Use the following SolidWorks Explorer tools to perform data management tasks, as follows:



<u>Preview</u> Displays an image of the selected part, assembly or drawing document.

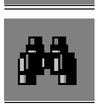


<u>Properties Summary</u> Lets you to enter or view summary information about the selected document.
<u>Custom Properties</u> Lets you specify custom properties for a document.
<u>Configuration Specific</u> Lets you specify configuration specific properties for the configurations of a part or assembly.

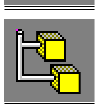


<u>Show References</u> Lists the references of any part (including derived or mirrored parts), assembly, or drawing document.

<u>Where Used</u> Lets you search for the all of the places where a specified part or assembly is used, including any derived or mirrored parts.



<u>Property Search</u> Lets you search for documents by referencing their properties.



<u>Edit Configurations</u> Lets you rename or delete a selected configuration in a part or assembly document, and updates its references.

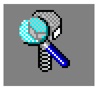
Edit Hyperlinks Lets you list and edit any hyperlinks that the selected document may have.

Options Lets you change the default settings of SolidWorks Explorer.

SolidWorks Explorer lets you easily perform the following tasks on a selected document (or documents), and updates references to and from the document appropriately. Right-click in the **Document** window to access these functions.

Copy
Rename
Replace

Preview



When you select an icon in the **Document** window, an image of the selected part, assembly or drawing document is displayed in the **Main** window.

Use the **Use full screen** check box to select the kind of display that you want.

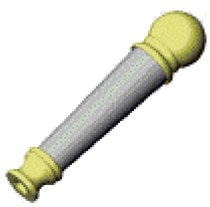
- When **Use full screen** is selected, a single image of the selected document fills the **Main** window.
- When **Use full screen** is not selected, multiple images can be displayed.

  Every time you select an icon in the **Document** window, the image for the corresponding part, assembly, or drawing is displayed.

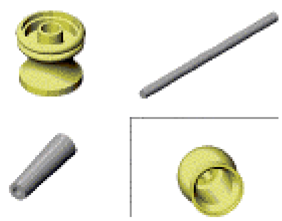
  A box appears around the image of the currently selected icon.

NOTE: An image is only available if the part was last saved independently; if the part was saved in the context of an assembly, an image of the part is not available.

Preview Example

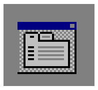


handle assembly



Components of handle assembly with the **Use full screen** check box cleared.

Custom Properties



Allows you to specify custom properties for the part, assembly, or drawing document, or configuration specific properties for the active part or assembly configuration.

You can use the properties in several ways:

- Custom columns in a Bill of Materials
- Advanced criteria for selection or show/hide
- Linked to text in notes

To enter custom properties:

1. Enter a **Name** for the property, or choose one from the list. If you plan to use the property as a custom column in a Bill of Materials, do not include any spaces in the **Name**.
2. Select the **Type** of value the property will use.
3. Enter a **Value** for the property that is compatible with the selection in the **Type** box.
4. Click **Add**.

   The **Properties** box displays the name, value and type of the property.
5. Click **Reset** to return to the previously saved properties.
6. Click **Apply** to save the changes.

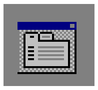
To modify custom properties:

1. Select the name in the **Properties** box.
2. Edit the **Type** or **Value** as needed.
3. Click **Modify**.

To delete custom properties:

1. Click the property in the **Properties** box that you want to delete.
2. Click **Remove**.

Properties Summary



Allows you to enter or view summary information about the part, assembly, or drawing document.

To enter or display summary information:

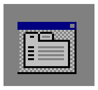
1. On the **Summary** tab, enter the appropriate information in the boxes provided for: **Author**, **Keywords**, **Comments**, **Title**, and **Subject**.

NOTE: If you insert a part identification number in the **Title** box, you can use that number automatically when you create a bill of materials in an assembly drawing that contains that part. See **Bill of Materials** for more information.

The system provides read-only **Statistics** about the document's creation date, the date last saved, and the name of the person who saved the document last.

2. Click **Reset** to return to the previously saved properties.
3. Click **Apply** to save the changes.

Configuration Specific



Allows you to specify configuration specific properties for the active part or assembly configuration.

You can use the properties in several ways:

- Custom columns in a Bill of Materials
- Advanced criteria for selection or show/hide
- Linked to text in notes

To enter custom properties:

1 Select a configuration from the **Configuration** list.

2 Enter a **Name** for the property, or choose one from the list. If you plan to use the property as a custom column in a Bill of Materials, do not include any spaces in the Name.

3 Select the **Type** of value the property will use.

4 Enter a **Value** for the property that is compatible with the selection in the **Type** box.

5 Click **Add**.

The **Properties** box displays the name, value and type of the property.

6 Click **Reset** to return to the previously saved properties.

7 Click **Apply** to save the changes.

To modify custom properties:

1 Select the name in the **Properties** box.

2 Edit the **Type** or **Value** as needed.

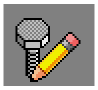
3 Click **Modify**.

To delete custom properties:

1 Click the property in the **Properties** box that you want to delete.

2 Click **Remove**.

Show References



You can list the references of any part (including derived parts), assembly, or drawing document.

To list the references for a part or a derived part, an assembly, an assembly component, or a drawing document:

Click the document's icon in the **Document** window.

The document is described by entries in the following columns:

- **File name** or **New name**. The name of the selected file appears in bold. Click the column heading to toggle between the display of the file name with the path or to display the file name without its path.
- **Size**. Displays the file size.
- **Type**. Displays the file type.
- **Modified**. Displays the time and date that the file was last modified.
- **Attributes**. Displays the attributes of the file, as reported by Windows Explorer.

To open the selected document, right-click in the **Main** window and select **Open file in SolidWorks**.

To export the list to an Excel spreadsheet, right-click in the **Main** window and select **Export list to Excel**.

NOTE: The first column of the Excel spreadsheet uses either the short or long version of the file name, depending on what is selected in the **File name** or **New name** column.

Show References Example

In this example, assume there is an assembly **A1** with two components **P1** and **P2**. **P2** has an in-context feature which references **P1** in the assembly **A1**.

When you show references for **A1**:

- **A1** appears in the **Main** window in bold font.
- **P1** and **P2** appear because they are components of the assembly.

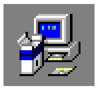
When you show references for **P1**:

P1 appears in the **Main** window in bold font.

When you show references for **P2**:

- **P2** appears in the **Main** window in bold font
- **A1** and **P1** appear because of the in-context feature.

Options



Lets you change the default settings of SolidWorks Explorer.

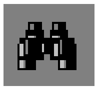
Add or edit a search path. Add new search paths or edit the existing search paths used by the **Where Used**, **Property Search**, and **Open** functions. A *search path* is the default drive and directory structure to search when locating files.

- **Search path:** **Modify**, **Delete**, or **Reset** the name of the search path, or type a new name then click **Add New** to add a new search path.
- **Directory list:** Select a directory in the **Directory list** to edit. If one or more directories are displayed, you can **Add New** folders, **Delete** existing folders, and **Move Up** or **Move Down** to change the search order.

Display: Select the **Show Description text at the bottom of every page** check box to see the message at the bottom of each SolidWorks Explorer page. This text changes to describe the function of each **Operation** icon as you select it.

File open: Select the **File Open should search 'Document Folders' for external references** check box to display the file's external references when you open a file from SolidWorks Explorer.

Property Search



Lets you search for documents by referencing their property pages. The documents that meet the specified search criteria are listed.

To define selection criteria and begin the search:

1 Select a **Search Path** from the list, or click **Browse** and select a folder to search in. Select the **Search subfolders** check box to include subfolders.

2 To use a set of previously specified criteria, click **Load criteria** and browse to the (**.sqy**) file.

3 Under **Define additional criteria**, select a **Property** from the list or type a user-defined name in the **Property** box.

4 Select a **Condition**, type a **Value**, and click **Add to list**.

The item appears in the **Criteria** box.

5 To add additional criteria for your search, click **And** or **Or**, then repeat steps **3** and **4**.

6 To delete a criteria from the list, click the item in the **Criteria** box, and click **Delete**. To clear the entire **Criteria** box, click **New Search**.

7 Click **Find Now** to begin the search.

When one or more documents are found that meet the specified criteria, they are listed with the following information:

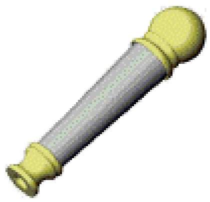
- **File name** or **New name**. Click the column heading to toggle between the display of the file name with the path or to display the file name without its path.
- **Type**. Displays the file type.
- **Size**. Displays the file size.
- **Date created**. Displays the time and date that the file was created.
- **Attributes**. Displays the attributes of the file, as reported by Windows Explorer.

8 To open a selected document, right-click a document in the **Main** window and select **Open file in SolidWorks**.

9 To export the list to an Excel spreadsheet, right-click and select **Export list to Excel**.

NOTE: The first column of the Excel spreadsheet uses either the short or long version of the file name, depending on what is selected in the **File name** or **New name** column.

Property Search Example



In the handle assembly shown, two of the components have a property called **Material**, with a value called **Brass**. To search for these components, use the following to define a criteria:

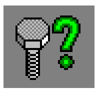
- **Property**. Material
- **Condition**. Is (exactly)
- **Value**. Brass

When you click **Find now**, the brass components are listed in the **Main** window.

Edit Hyperlinks

Select an icon in the **Document** window to list and edit any hyperlinks that it may have.To open a hyperlink in the selected document, double-click the hyperlink text or icon.

Where Used



Lets you search for the all of the places where a specified part or assembly is used, including all the assembly or drawing documents, or any derived or mirrored parts.

To find where a document is used:

1. Select a **Search Path** from the list, or click **Browse** and select a folder to search in. The name of the selected path appears in the **Look in** box.
2. In the **Document** window, click the icon of the document that you want to use as the subject of your search. The name of the document appears in the **File** box.
3. Set the search criteria by selecting from the following check boxes:
   - **Look for assemblies and drawings**
   - **Look for derived/mirrored parts**
   - **Look for models defined in this assembly**
   - **Search subfolders**
4. Click **Find Now** to begin the search.
5. Click **New Search** to clear the selections from a previous search and start again.

   The documents are described by entries in the following columns:

   - **Used by**. Lists the other documents that use the subject of the search. To display the full path name and file name, click the column heading when it says **click to show long name**.

     To display the file name without its path, click the column heading when it says **click to show short name**.
   - **Used as**. Describes the way the search subject is used as follows: **model reference**, **modeling context**, or **derived/mirrored part**.
   - **Found**. Lists all the occurrences of the document that is the search subject. To display the full path name of its various locations, click the column heading when it says **click to show long name**.

To open a selected document, right-click a document in the **Main** window and select **Open file in SolidWorks**.

To export the list to an Excel spreadsheet, right-click and select **Export list to Excel**.

NOTE: The columns of the Excel spreadsheet use either the short or long version of the file name, depending on what is selected in the **Used by** or **Found** columns of the list in the **Where Used** dialog.

Where Used Example

In this example, assume there is an assembly **A1** with two components **P1** and **P2**. **P1** and **P2** each have their own detail drawing named **D1** and **D2** respectively.

- When you search for where **P1** is used, **A1** and **D1** appear in the **Main** window.
- When you search for where **P2** is used, **A1** and **D2** appear in the **Main** window.File

To open a file in the SolidWorks Explorer:

- Click **File** on the menu bar and browse to select a file to open.
- Click **Browse** next to the **Path** to locate a document, then click **Open**.
- Drag a document from Windows Explorer to the **Document** window.

Edit

To copy or rename a file from the SolidWorks Explorer:

- Click **Edit** on the menu bar.

  - or -

- Right-click the file name in the **Document** window and select **Copy**, **Rename**, or **Replace**, if appropriate.

The **Copy Document**, **Rename Document**, or **Replace Document** dialog box appears in the **Main** window. You are able to browse to find the filename that you want, if necessary.

Tools

When you click **Tools** on the menu bar, you can select any of the SolidWorks Explorer operations from the list:

Preview	**Property Search**
Properties	**Edit Configurations**
Show References	**Edit Hyperlinks**
Where Used	**Options**

Copy

Copies a selected document (or a document and its children) and updates references.

To copy a document:

1 Right-click a document in the **Document** window and select **Copy**.

The document's name appears in the **Copy** box.

The new document name and suggested path appear in the **To** box. The new name is the same as the original name with the addition of "Rev" and a revision number. For example: "Part1.SLDPRT" becomes "Part1Rev2.SLDPRT"

NOTE: It is important to remember that SolidWorks Explorer does <u>not</u> provide true revision control, in the same manner as a PDM (Product Data Management) tool.

2 To copy the document to a new location, click **Browse** and select a new folder.

3 If the document has child documents that are dependent on it, you can select the **Copy children** check box to copy them all at the same time and to the same location.

If you select **Copy children**, the document's dependencies are listed. You have several options:

- Instead of Rev(*n*), you can enter other revision identifying characters to add to the original document name.
- You can select **Prefix** or **Suffix** to change the location of the revision identifier in relation to the original document name.
  For example: "Rev2Part1.SLDPRT" or "Part1Rev2.SLDPRT"
- You can make copies of all the documents in the list or you can clear the check marks of some documents so copies of them will not be made. Click **Deselect all** to clear all of the check marks.

4 You can search for other places where this document is used by selecting the **Find where used** check box, clicking **Search rules** to select search criteria to use, then clicking **Find now**. Click **Reset** if you want to perform a new search.

- **<u>Look for assemblies and drawings</u>**
- **<u>Look for derived/mirrored parts</u>**
- **<u>Look for models defined in this assembly</u>**
- **<u>Search subfolders</u>**

5 In the **Find where used** list you can do the following:

- To open a selected document, right-click a document and select **Open File in SolidWorks**.
- To export the list to an Excel spreadsheet, right-click and select **Export list to Excel**.

NOTE: The first column of the Excel spreadsheet uses either the short or long version of the file name, depending on what is selected in the **Used by** and **New name** columns in the **Find where used** list.

6 Click **Apply** to make the copies of the selected documents.

The names of the documents that were copied successfully are displayed in blue font. The names of the documents that were not copied successfully are displayed in red font.

Rename

Renames one or more selected documents and updates all the references.

To rename a document:

1 Right-click a document in the **Document** window and select **Rename**.

The document's name appears in the **Rename** box and in the **To** box.

2 Select the name in the **To** box and enter a new name.

3 To search for other places where this document is used select the **Find where used** check box. Click **Search rules** to select search criteria to use, and then click **Find Now**. Click **Reset** if you want to perform a new search.

- **Look for assemblies and drawings**
- **Look for derived/mirrored parts**
- **Look for models defined in this assembly**
- **Search subfolders**

4 In the resulting list you can do the following:

- If you leave the **Update** check box checked, all the documents in the list are automatically updated to reference the renamed document by its new name.
  If you clear the **Update** check box, you must manually update each of the referencing documents yourself.
- To open a selected document, right-click a document and select **Open file in SolidWorks**.
- To export the list to an Excel spreadsheet, right-click and select **Export list to Excel**.

NOTE: The first column of the Excel spreadsheet uses either the short or long version of the file name, depending on what is selected in the **Used by** list.

5 Click **Apply** to change the name of the selected documents.

The names of the documents that were changed and updated successfully are displayed in blue font. The names of the documents that were not changed successfully are displayed in red font.

Replace

Replaces a selected part or assembly document and updates its references.

To replace a document:

1 Right-click a part or assembly **Document** in the document window and select **Replace**.

The document's name appears in the **Replace** box and in the **With** box.

2 Select the name in the **With** box and click **Browse** to find a document to use as a replacement.

3 To search for other places where this document is used select the **Find where used** check box. Click **Search rules** to select search criteria to use, and then click **Find now**. Click **Reset** if you want to perform a new search.

- **Look for assemblies and drawings**
- **Look for derived/mirrored parts**
- **Look for models defined in this assembly**
- **Search subfolders**

4 In the resulting list you can do the following:

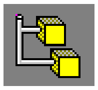
- If you leave the **Update** check box checked, all the documents in the list are automatically updated to reference the replacement document by its name.
  If you clear the **Update** check box, you must manually update each of the referencing documents yourself.
- To open a selected document, right-click a document and select **Open file in SolidWorks**.
- To export the list to an Excel spreadsheet, right-click and select **Export list to Excel**.

NOTE: The first column of the Excel spreadsheet uses either the short or long version of the file name, depending on what is selected in the **Used by** column in the **Replace Document** list.

5 Click **Apply** to replace the selected documents.

The names of the documents that were replaced and updated successfully are displayed in blue font. The names of the documents that were not replaced successfully are displayed in red font.

Edit Configurations



Renames or deletes a selected configuration in part or assembly document, and updates its references. The part or assembly document must not be open in either SolidWorks or the SolidWorks Explorer.

To rename a configuration:

1 Select a document's icon in the **Document** window to list the configurations of the selected document.

2 Select the configuration to rename.

3 Click-pause-click the configuration name and enter a new name.

The current configuration name and the new name are displayed.

4 Click **Apply**.

If the document is not open, the name change takes place.

If the document is in use, the name cannot change. Click **Reset** and make the name change at another time.

To delete a configuration:

1 Select a document's icon in the **Document** window to list the configurations of the selected document.

2 Select the configuration to delete and press the **Delete** key.

3 Click **Apply**.

If the document is not open, the deletion takes place.

If the document is in use, you cannot delete the configuration. Click **Reset** and delete the configuration at another time.

Look for assemblies and drawings

Searches for all assemblies and drawings where the selected file is used.

Look for derived/mirrored parts

Searches for all dervied parts (base, mirror, or derived component) where the selected file is used.

Look for models defined in this assembly

Searches for all components created in the context of the selected assembly.

Search subfolders

Searches subfolders for documents where the selected document is used.

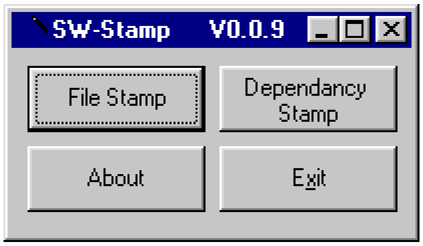
Importing/Exporting SolidWorks Documents

You can import files to the SolidWorks software from other applications. You can export SolidWorks documents to a number of formats for use with other applications. The following table displays the data translation methods available for SolidWorks documents:

	Parts		Assemblies		Drawings	
	Import	Export	Import	Export	Import	Export
IGES	X	X	X	X		
Parasolid	X	X	X	X		
STEP	X	X	X	X		
ACIS	X	X	X	X		
VDAFS	X	X				
STL		X		X		
DXF/DWG					X	X
VRML	X	X	X	X		
TIFF	X	X	X	X		X
Unigraphics	X					
Pro/ENGINEER	X					

Plot Stamp Utility

A free SolidWorks plot stamp utility is available from Contract CADD Group's webstite at: http://www.contractcaddgroup.com/download/VB/sw-stamp.zip



The plot stamp program places a plot stamp in a SolidWorks 2000 drawing. The plot stamp program also includins an option to place a stamp of all files used to create the drawing (reference files including all parts in all referenced sub assemblies.)

Example: