SolidWorks® 2006

SolidWorks Essentials
Parts and Assemblies

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Lesson 1
Introduction

Upon successful completion of this lesson, you will be able to:

- Describe the key characteristics of a feature-based, parametric solid modeler.
- Distinguish between sketched and applied features.
- Identify the principal components of the SolidWorks user interface.
- Explain how different dimensioning methodologies convey different design intents.

About This Course

The goal of this course is to teach you how to use the SolidWorks mechanical design automation software to build parametric models of parts and assemblies and how to make simple drawings of those parts and assemblies.

SolidWorks 2006 is such a robust and feature rich application that it is impractical to cover every minute detail and aspect of the software and still have the course be a reasonable length. Therefore, the focus of this course is on the fundamental skills and concepts central to the successful use of SolidWorks 2006. You should view the training course manual as a supplement to, not a replacement for, the system documentation and on-line help. Once you have developed a good foundation in basic skills, you can refer to the on-line help for information on less frequently used command options.

Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Experience with the Windows™ operating system.
- Read the *Introducing SolidWorks* manual. A hardcopy of this manual is included with your software. Or, you can access an online version of this manual by clicking **Help, Introducing SolidWorks**.
- Completed the online tutorials that are integrated in the SolidWorks software. You can access the online tutorials by clicking **Help, Online Tutorial**.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. Rather than focus on individual features and functions, a process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a design task.

Using this Book

This training manual is intended to be used in a classroom environment under the guidance of an experienced SolidWorks instructor. It is not intended to be a self-paced tutorial. The examples and case studies are designed to be demonstrated “live” by the instructor.

Laboratory Exercises

Laboratory exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration portion of the course. They are designed to represent typical design and modeling situations while being modest enough to be completed during class time. You should note that many students work at different paces. Therefore, we have included more lab exercises than you can reasonably expect to complete during the course. This ensures that even the fastest student will not run out of exercises.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that complements this objective.

About the CD

Bound inside the rear cover is a CD containing copies of the various files that are used throughout this course. They are organized by lesson number. The `Case Study` folder within each lesson contains the files your instructor uses while presenting the lessons. The `Exercises` folder contains any files that are required for doing the laboratory exercises.

Windows® 2000 and Windows® XP

The screen shots in this manual were made using SolidWorks 2006 running on Windows® 2000 and Windows® XP. You may notice differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Conventions Used in this Book

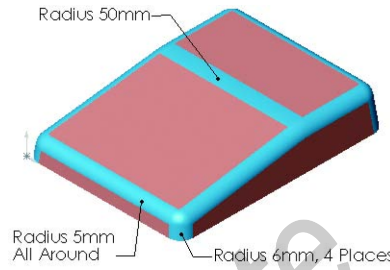
This manual uses the following typographic conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, **Insert, Boss** means choose the **Boss** option from the **Insert** menu.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

Use of Color

The SolidWorks user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of the SolidWorks software. To take maximum advantage of this, the training manuals are printed in full color.

Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of a filleting operation with the fillets in a different color, even though by default, the SolidWorks software would not display the results in that way.



What is the SolidWorks Software?

SolidWorks mechanical design automation software is a *feature-based*, *parametric solid modeling* design tool which takes advantage of the easy to learn Windows™ graphical user interface. You can create *fully associative* 3-D solid models with or without *constraints* while utilizing automatic or user defined relations to capture *design intent*.

The italicized terms mean:

- **Feature-based**
  Just as an assembly is made up of a number of individual piece parts, a SolidWorks model is also made up of individual constituent elements. These elements are called features.

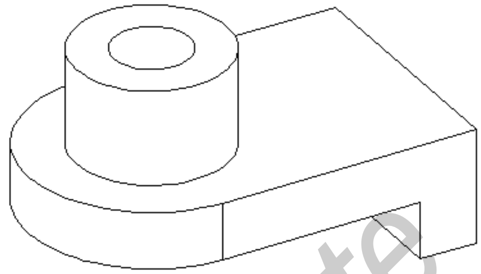
  When you create a model using the SolidWorks software, you work with intelligent, easy to understand geometric features such as bosses, cuts, holes, ribs, fillets, chamfers, and draft. As the features are created they are applied directly to the work piece.

  Features can be classified as either sketched or applied.

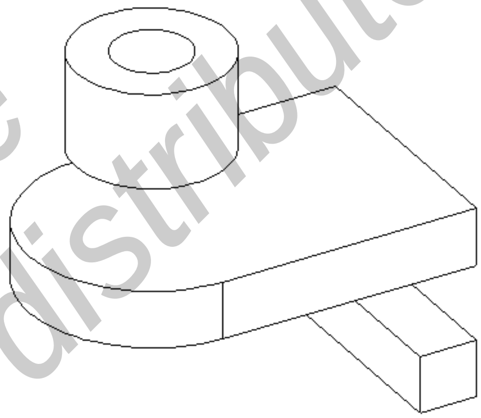
  - Sketched Features: One that is based upon a 2-D sketch. Generally that sketch is transformed into a solid by extrusion, rotation, sweeping or lofting.
  - Applied Features: Created directly on the solid model. Fillets and chamfers are examples of this type of feature.

  The SolidWorks software graphically shows you the feature-based structure of your model in a special window called the FeatureManager® design tree. The FeatureManager design tree not only shows you the sequence in which the features were created, it gives you easy access to all the underlying associated information. You will learn more about the FeatureManager design tree throughout this course.

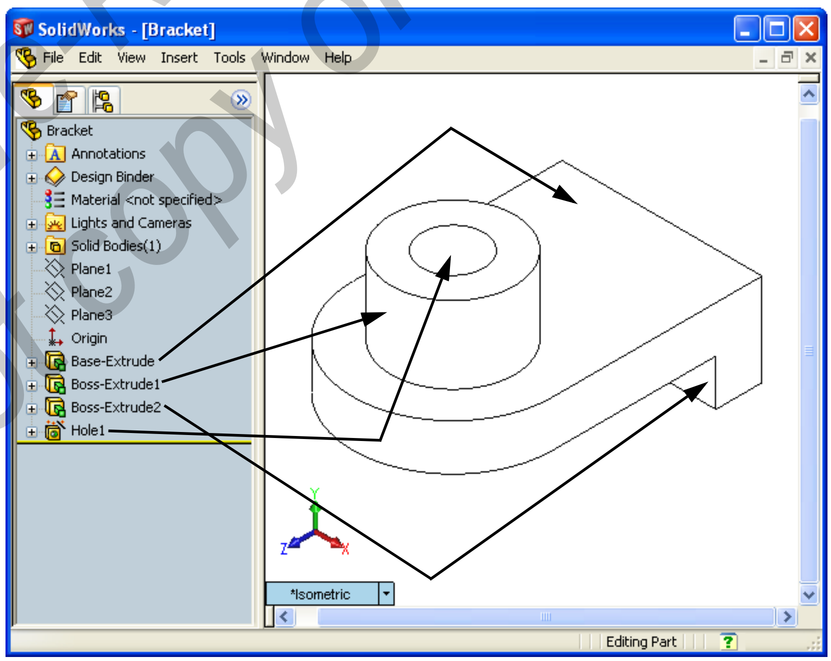
To illustrate the concept of feature-based modeling, consider the part shown at the right:



This part can be visualized as a collection of several different features – some of which add material, like the cylindrical boss, and some which remove material, like the blind hole.



If we were to map the individual features to their corresponding listing in the FeatureManager design tree, it would look like this:



- **Parametric**
  The dimensions and relations used to create a feature are captured and stored in the model. This not only enables you to capture your design intent, it also allows you to quickly and easily make changes to the model.
  - Driving Dimensions: These are the dimensions used when creating a feature. They include the dimensions associated with the sketch geometry, as well as those associated with the feature itself. A

simple example of this would be a feature like a cylindrical boss. The diameter of the boss is controlled by the diameter of the sketched circle. The height of the boss is controlled by the depth to which that circle was extruded when the feature was made.

- Relations: These include such information as parallelism, tangency, and concentricity. Historically, this type of information has been communicated on drawings via feature control symbols. By capturing this in the sketch, SolidWorks enables you to fully capture your design intent up front, in the model.

- **Solid Modeling**
  A solid model is the most complete type of geometric model used in CAD systems. It contains all the wire frame and surface geometry necessary to fully describe the edges and faces of the model. In addition to the geometric information, it has the information called topology that relates the geometry together. An example of topology would be which faces (surfaces) meet at which edge (curve). This intelligence makes operations such a filleting as easy as selecting an edge and specifying a radius.
- **Fully Associative**
  A SolidWorks model is fully associative to the drawings and assemblies that reference it. Changes to the model are automatically reflected in the associated drawings and assemblies. Likewise, you can make changes in the context of the drawing or assembly and know that those changes will be reflected back in the model.
- **Constraints**
  Geometric relationships such as parallel, perpendicular, horizontal, vertical, concentric, and coincident are just some of the constraints supported in SolidWorks. In addition, equations can be used to establish mathematical relationships among parameters. By using constraints and equations, you can guarantee that design concepts such as through holes or equal radii are captured and maintained.
- **Design Intent**
  Design intent is your plan as to how the model should behave when it is changed. For example, if you model a boss with a blind hole in it, the hole should move when the boss is moved. Likewise, if you model a circular hole pattern of six equally spaced holes, the angle between the holes should change automatically if you change the number of holes to eight. The techniques you use to create the model determine how and what type of design intent you capture.

Design Intent

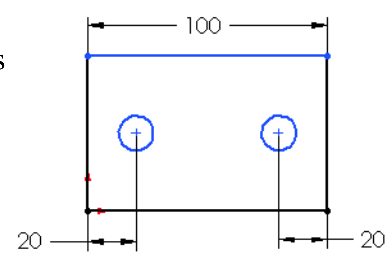
In order to use a parametric modeler like SolidWorks efficiently, you must consider the design intent before modeling. Design intent is your plan as to how the model should behave when it is changed. The way in which the model is created governs how it will be changed. Several factors contribute to how you capture design intent:

- **Automatic (sketch) Relations**
  Based on how geometry is sketched, these relations can provide common geometric relationships between objects such as parallel, perpendicular, horizontal, and vertical.
- **Equations**
  Used to relate dimensions algebraically, they provide an external way to force changes.
- **Added Relations**
  Added to the model as it is created, relations provide another way to connect related geometry. Some common relations are concentric, tangent, coincident, and collinear.
- **Dimensioning**
  The way in which a sketch is dimensioned will have an impact upon is design intent. Add dimensions in a way that reflects how you would like to change them.

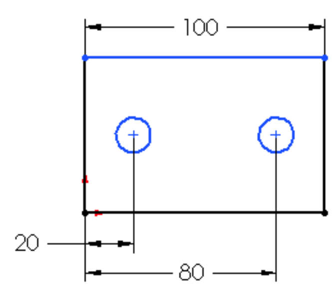
Examples of Design Intent

Some examples of different design intent in a sketch are shown below.

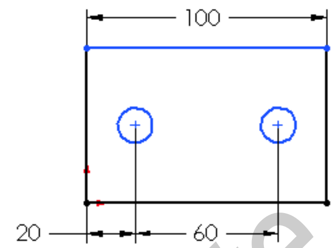
A sketch dimensioned like this will keep the holes **20mm** from each end regardless of how the overall plate width, **100mm**, is changed.



Baseline dimensions like this will keep the holes positioned relative to the left edge of the plate. The positions of the holes are not affected by changes in the overall width of the plate.



Dimensioning from the edge and from center to center will maintain the distance between the hole centers and allow it to be changed that way.

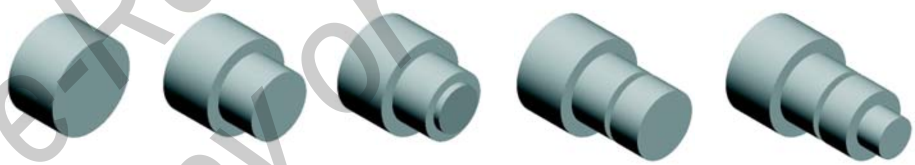


How Features Affect Design Intent

Design intent is affected by more than just how a sketch is dimensioned. The choice of features and the modeling methodology are also important. For example, consider the case of a simple stepped shaft as shown at the right. There are several ways a part like this could be built.

The "Layer Cake" Approach

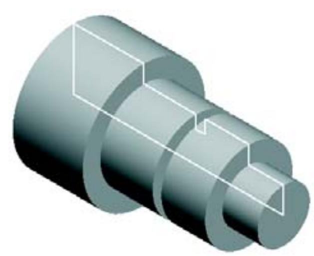
The layer cake approach builds the part one piece at a time, adding each layer, or feature, onto the previous one, like this:



Changing the thickness of one layer has a ripple effect, changing the position of all the other layers that were created after it.

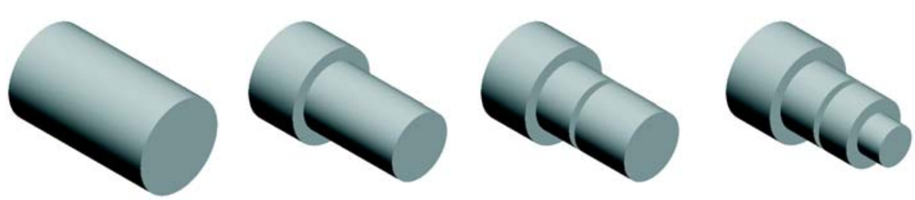
The "Potter's Wheel" Approach

The potter's wheel approach builds the part as a single, revolved feature. A single sketch representing the cross section includes all the information and dimensions necessary to make the part as one feature. While this approach may seem very efficient, having all the design information contained within a single feature limits flexibility and can make changes awkward.



The Manufacturing Approach

The manufacturing approach to modeling mimics the way the part would be manufactured. For example, if this stepped shaft was turned on a lathe, you would start with a piece of bar stock and remove material using a series of cuts.



Unselectable Icons

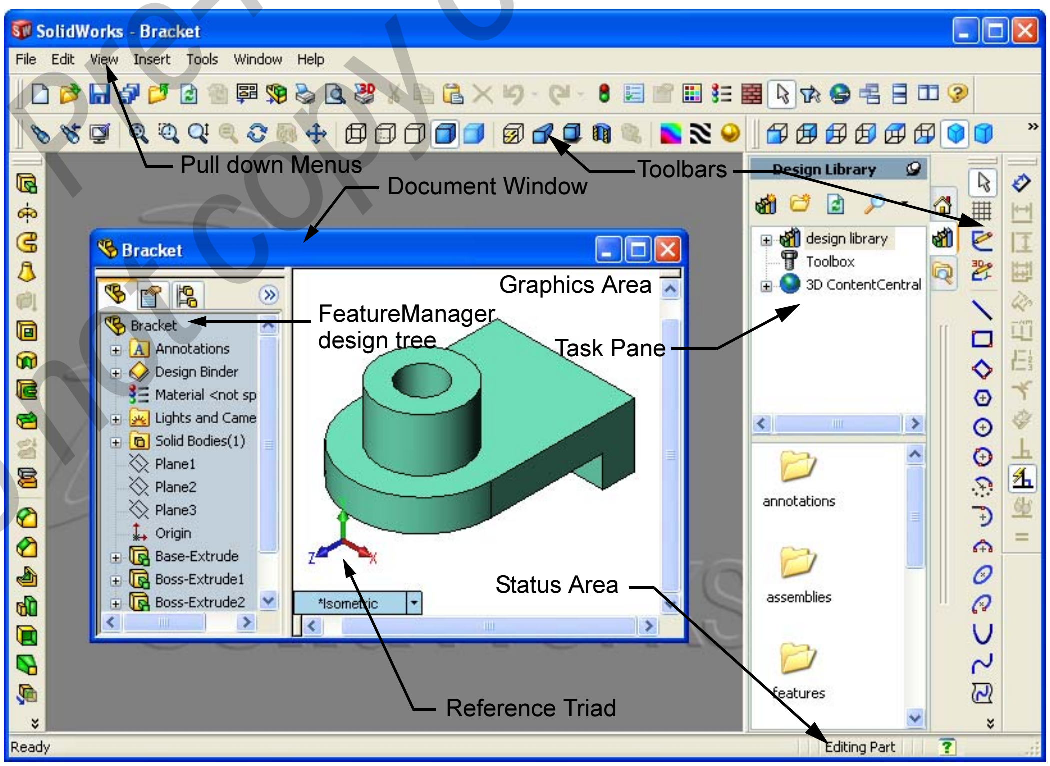
At times you will notice commands, icons, and menu options that are grayed out and unselectable. This is because you may not be working in the proper environment to access those options. For example, if you are working in a sketch (**Edit Sketch** mode), you have full access to all the sketch tools. However, you cannot select the icons such as fillet or chamfer on the Features toolbar. Likewise, when you are working in the **Edit Part** mode, you *can* access these icons but the sketch tools are grayed out and unselectable. This design helps the inexperienced user by limiting the choices to only those that are appropriate, graying out the inappropriate ones.

To Pre-select or Not?

As a rule, the SolidWorks software does not require you to pre-select objects before opening a menu or dialog box. For example, if you want to add some fillets to the edges of your model, you have complete freedom – you can select the edges first and then click the **Fillet** tool or you can click the **Fillet** tool and then select the edges. The choice is yours.

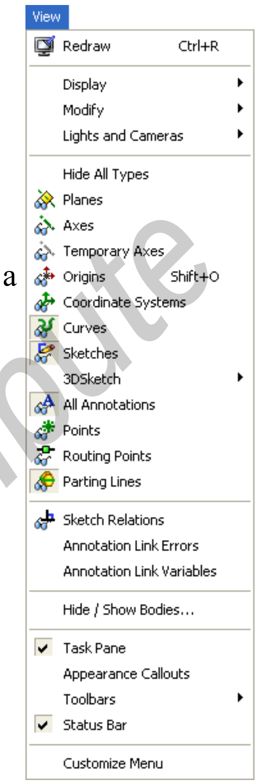
The SolidWorks User Interface

The SolidWorks user interface is a native Windows interface, and as such behaves in the same manner as other Windows applications. Some of the more important aspects of the interface are identified below.



Menus

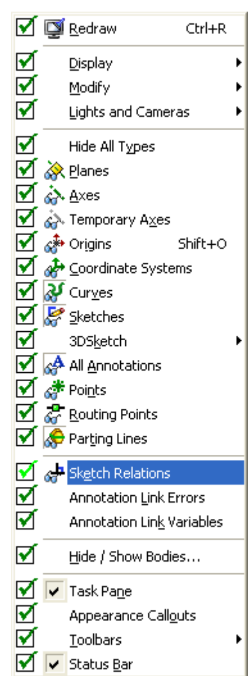
Menus provide access to all the commands that the SolidWorks software offers.



When a menu item has a right-pointing arrow like this: Display ▸, it means there is a sub-menu associated with that choice.

When a menu item is followed by a series of dots like this: Orientation..., it means that option opens a dialog box with additional choices or information.

When the **Customize Menu** item is selected, each item appears with a check box. Clearing the check box removes the associated item from the menu.



Keyboard Shortcuts

Some menu items indicate a keyboard shortcut like this: Redraw Ctrl+R. SolidWorks conforms to standard Windows conventions for such shortcuts as **Ctrl+O** for **File, Open**; **Ctrl+S** for **File, Save**; **Ctrl+Z** for **Edit, Undo** and so on. In addition, you can customize SolidWorks by creating your own shortcuts.

Toolbars

The toolbar menus provide shortcuts enabling you to quickly access the most frequently used commands. The toolbars are organized according to function and you can customize them, removing or rearranging the tools according to your preferences. The individual options on them will be covered in detail throughout this course.

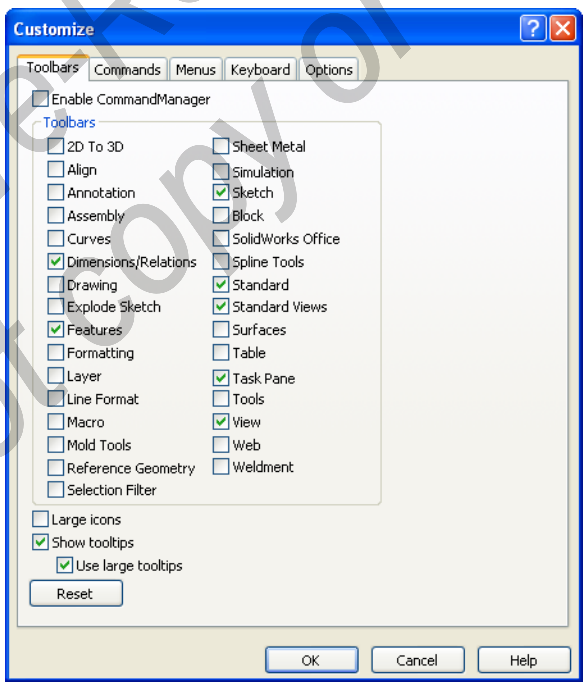
Example of a Toolbar

An example of a toolbar, in this case the Standard toolbar, is shown below. This toolbar contains such commonly used functions as opening new or existing documents, saving documents, printing, copying and pasting objects, undo, redo, and help.

Making Toolbars Visible

You can turn toolbars on or off using one of three methods:

- **Click Tools, Customize.**
  On the **Toolbars** page, click the check boxes to select each toolbar you want to display. Clear the check boxes of the toolbars you want to hide.

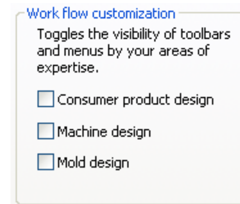


Note

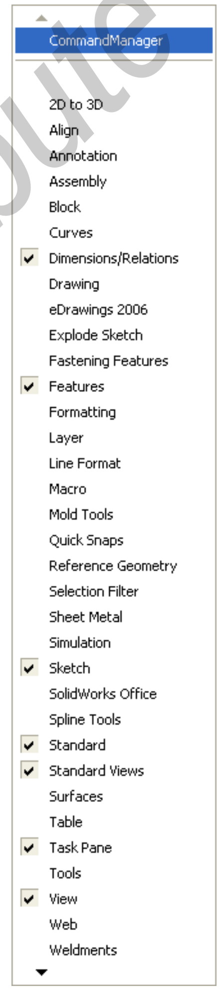
In order to access **Tools, Customize**, you must have a document open. Also the **Commands** tab can be used to add or remove icons from toolbars.

Work flow Customization

Toolbars can be turned on and off by industry using **Work flow customization** on the **Options** tab. Several industries are available.

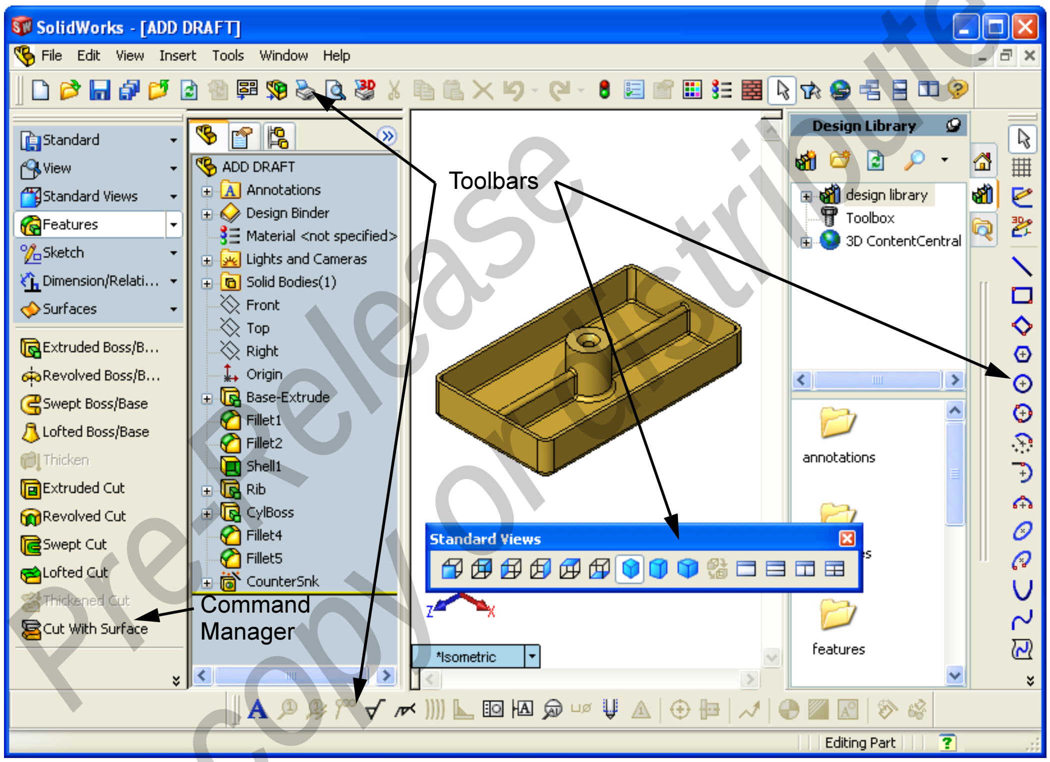


- **Right-click in the toolbar area of the SolidWorks window.**
  Check marks indicate which toolbars are currently visible. Clear the check marks of the toolbars you want to hide.
- **Click View, Toolbars.**
  This displays the same list of toolbars.



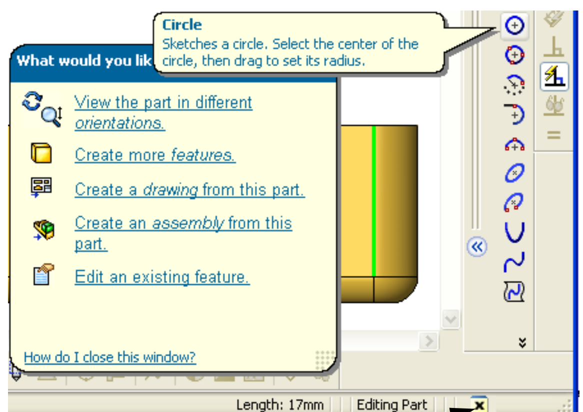
Arranging the Toolbars

The toolbars, including the Command Manager, can be arranged in many ways. They can be docked around all four borders of the SolidWorks window or dragged onto the graphics or FeatureManager areas. These positions are “remembered” when you exit SolidWorks so the next time you start SolidWorks, the toolbars will be where you left them. One such arrangement is shown below.



Quick Tips

Quick Tips are part of the on-line help system. They ask “What would you like to do?” and provide typical answers based on the current task. Clicking an answer highlights the toolbar and icon required to perform that task.



Toggle Quick Tips On/Off

FeatureManager Design Tree

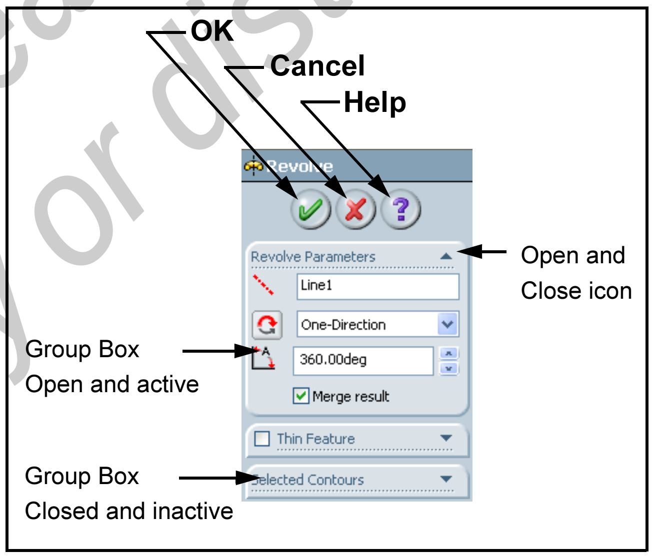
The FeatureManager design tree is a unique part of the SolidWorks software that visually displays all the features in a part or assembly. As features are created they are added to the FeatureManager design tree. As a result, the FeatureManager design tree represents the chronological sequence of modeling operations. The FeatureManager design tree also allows access to the editing of the features (objects) that it contains.

PropertyManager Menus

Many SolidWorks commands are executed through PropertyManager menus. PropertyManager menus occupy the same screen position as the FeatureManager design tree and replace it when they are in use.

The color scheme and appearance of the PropertyManager menus can be modified through **Tools, Options, Colors**. See the SolidWorks online help for more information.

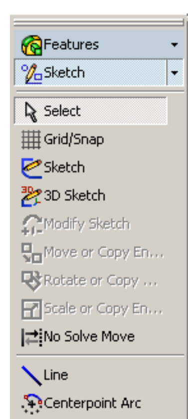
The top row of buttons contains the standard **OK**, **Cancel** and **Help** buttons.

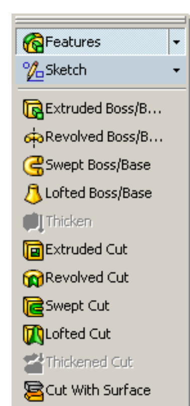


Below the top row of buttons are one or more **Group Boxes** that contain related options. They can be opened (expanded) or closed (collapsed) and in many cases made active or inactive.

A Word about the Command Manager

The **Command Manager** is a set of toolbars geared towards helping the novice user, working alone, to perform specific tasks. For example, the part version of the toolbar has two main groupings: **Features** and **Sketches** listed as buttons on the top.

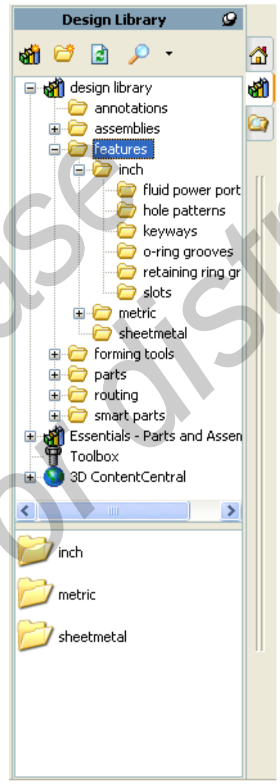


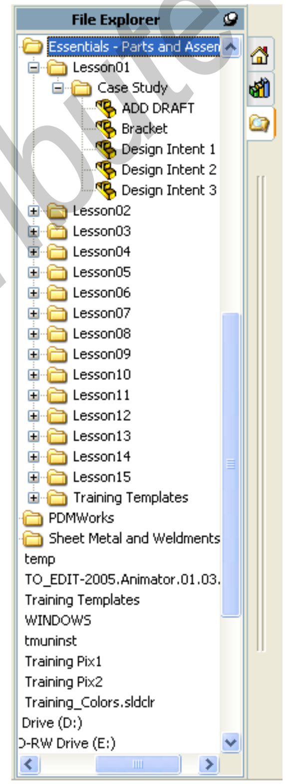


This manual will *not* use the Command Manager toolbar. It will instead use the more general standard toolbar set. For more information see *Toolbars* on page 12.

Taskpane

The **Task Pane** window is used to house the **SolidWorks Resources**, **Design Library** and **File Explorer** options. The window appears on the right by default but it can be moved and resized. It can be opened/closed, tacked or moved from its default position on the right side of the interface.

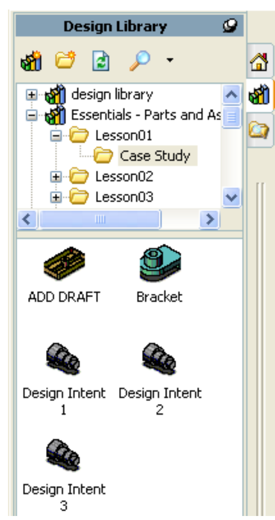




Opening Labs with the Design Library

You can open parts and assemblies required for lab exercises using the design library. Add the class files to the design library using this procedure.

- Open the **Task Pane** and the **Design Library**.
- Click **Add File Location**.
- Select the `Essentials - Parts and Assemblies` folder used for the class files. It should be found under the `SolidWorks 2006 Training Files` folder.
- Click **OK**.



Double-click the icon of the part or assembly in the **Design Library** to open it.

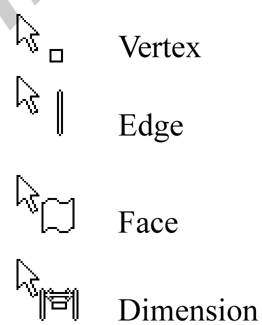
Mouse Buttons

The left, right and middle mouse buttons have distinct meanings in SolidWorks.

- **Left**
  Select objects such as geometry, menus buttons and objects in the FeatureManager design tree.
- **Right**
  Activates a context sensitive shortcut menu. The contents of the menu differ depending on what object the cursor is over. These menus also represent shortcuts to frequently used commands.
- **Middle**
  Dynamically rotates, pans or zooms a part or assembly. Pans a drawing.

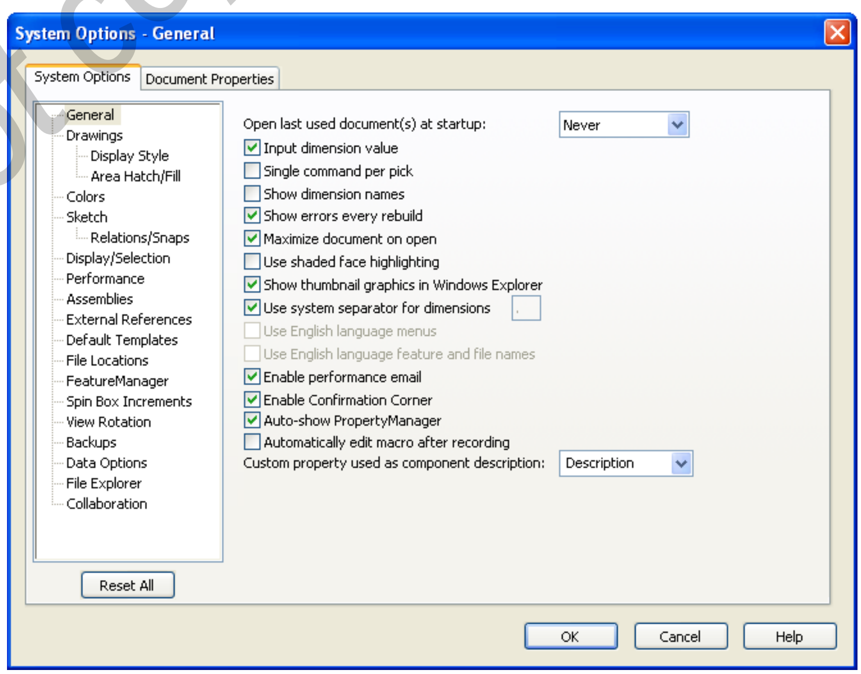
System Feedback

Feedback is provided by a symbol attached to the cursor arrow indicating what you are selecting or what the system is expecting you to select. As the cursor floats across the model, feedback will come in the form of symbols, riding next to the cursor. The illustration at the right shows some of the symbols: vertices, edges, faces and dimensions.



Options

Located on the **Tools** menu, the **Options** dialog box allows you to customize the SolidWorks software to reflect such things as your company's drafting standards as well as your individual preferences and work environment.



Customization

You have several levels of customization. They are:

- **System options**
  The options grouped under the heading **System Options** are saved on your system and affect every document you open in your SolidWorks session. System settings allow you to control and customize your work environment. For example, you might like working with colored viewport background. I don't. Since this is a system setting, parts or assemblies opened on your system would have a colored viewport. The same files opened on my system would not.
- **Document properties**
  Certain settings are applied to the individual document. For example, units, drafting standards, and material properties (density) are all document setting. They are saved with the document and do not change, regardless of whose system the document is opened on.

  For more information about the options settings that are used in this course, refer to *Options Settings* on page 455 in the Appendix.
- **Document templates**
  Document templates are pre-defined documents that were set up with certain specific settings. For example, you might want two different templates for parts. One with English settings such as ANSI drafting standards and inch units, and one with metric settings such as millimeters units and ISO drafting standards. You can set up as many different document templates as you need. They can be organized into different folders for easy access when opening new documents. You can create document templates for parts, assemblies, and drawings.

  For more detailed instructions on how to create document templates, refer to *Document Templates* on page 455 in the Appendix.
- **Object**
  Many times the properties of an individual object can be changed or edited. For example, you can change the default display of a dimension to suppress one or both extension lines, or you can change the color of a feature.

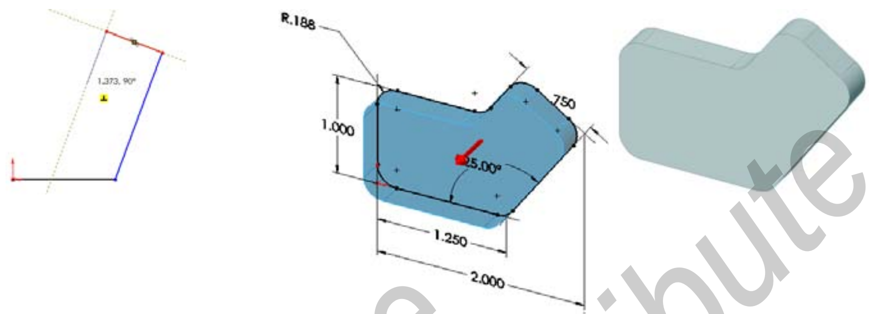
Lesson 2
Introduction to Sketching

Upon successful completion of this lesson, you will be able to:

- Create a new part.
- Insert a new sketch.
- Add sketch geometry.
- Establish sketch relations between pieces of geometry.
- Understand the state of the sketch.
- Use sketch tools to add chamfers and fillets.
- Extrude the sketch into a solid.

2D Sketching

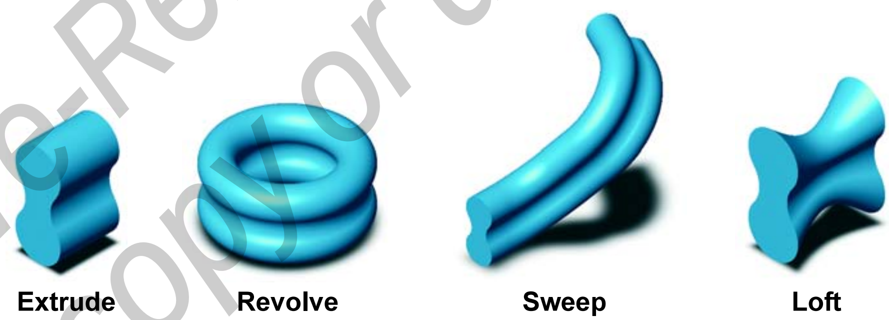
This lesson introduces 2D sketching, the basis of modeling in SolidWorks.



Sketches are used for all sketched features in SolidWorks including:

- Extrusions
- Sweeps
- Revolves
- Lofts

The illustration below shows how a given sketch can form the basis of several different types of features.



In this lesson, only extruded features will be covered. The others will be covered in detail in later lessons or courses.

Stages in the Process

Every sketch has several characteristics that contribute to its shape, size and orientation.

- **New part**
  New parts can be created in inch, millimeter or other units. Parts are used to create and hold the solid model.
- **Sketches**
  Sketches are collections of 2D geometry that are used to create solid features.
- **Sketch geometry**
  Types of 2D geometry such as lines, circles and rectangles that make up the sketch.
- **Sketch relations**
  Geometric relationships such as horizontal and vertical are applied to the sketch geometry. The relations restrict the movement of the entities.

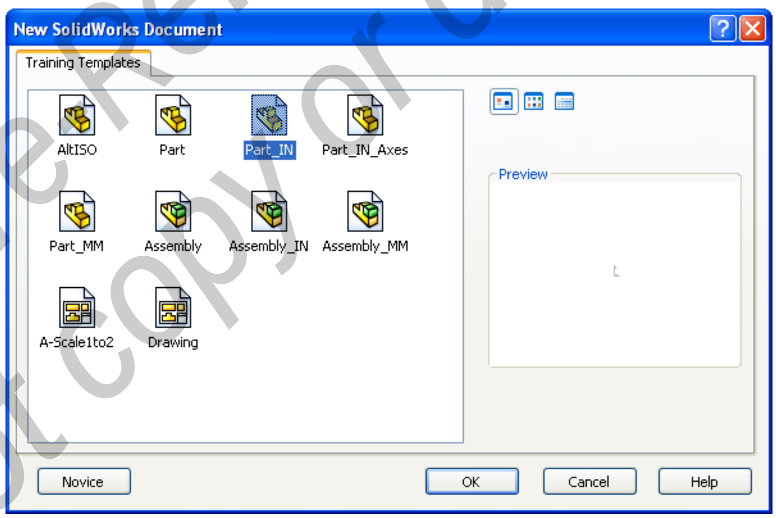
- **State of the sketch**
  Each sketch has a status that determines whether it is ready to be used or not. The state can be fully-, under- or over defined.
- **Sketch tools**
  Tools can be used to modify the sketch geometry that has been created. This often involves the trimming or extension of the entities.
- **Extruding the sketch**
  Extruding uses the 2D sketch to create a 3D solid feature.

Procedure

The process in this lesson includes sketching and extrusions. To begin with, a new part file is created.

1 New part.

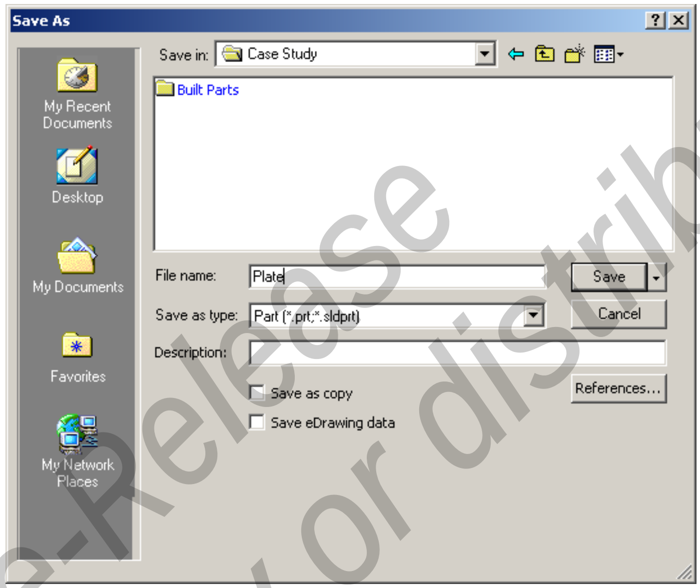
Click **New**, or click **File, New** on the Standard toolbar. Click the `Part_IN` template from the **Training Templates** tab on the **New SolidWorks Document** dialog box, and click **OK**.



The part is created with the settings of the template. One key setting is the part's units. As the name implies, this part template uses inches as the units. You can create and save any number of different templates, all with different settings.

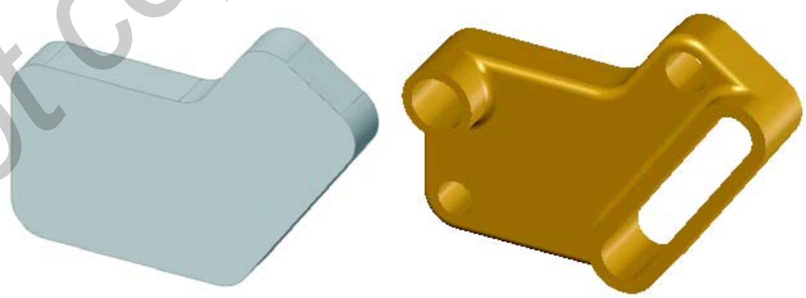
2 Filing a part.

Using the **Save** option from the **File** menu or selecting the **Save** button on the Standard toolbar, file the part under the name `Plate`. The extension, `*.sldprt`, is added automatically. Click **Save**.



What are We Going to Sketch?

The first feature of a part will be created in this section. That initial feature is just the first of many features needed to complete the part.



Sketching

Sketching is the act of creating a 2-dimensional profile comprised of wireframe geometry. Typical geometry types are lines, arcs, circles and ellipses. Sketching is dynamic, with feedback from the cursor to make it easier.

Default Planes

To create a sketch, you must choose a plane on which to sketch. The system provides three initial planes by default. They are `Front`, `Top`, and `Right`.

Introducing: Insert Sketch

When creating a new sketch, **Insert Sketch** opens the sketcher on the currently selected plane or planar face. You also use **Insert Sketch** to edit an existing sketch.

You must select a reference plane or a planar face of the model after clicking **Insert, Sketch**. The cursor appears indicating that you should select a face or plane.

Where to Find It

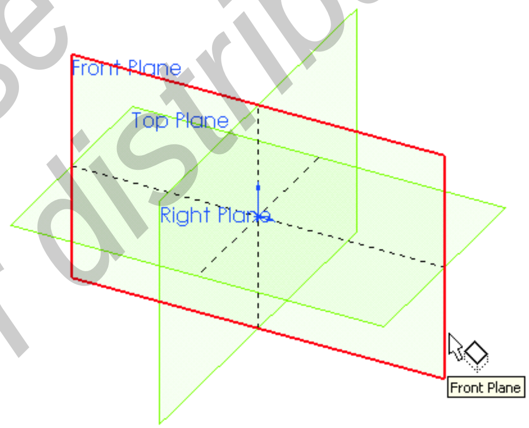
You can access the **Insert Sketch** command in several ways.

- On the Sketch toolbar click the tool.
- Or, on the **Insert** menu, click **Sketch**.
- Or, with the cursor positioned over a planar face or plane of the model, right-click and choose **Insert Sketch** from shortcut menu.

---

3 Open a new sketch.

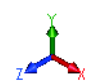
Open the sketch by either clicking or choosing **Sketch** from the **Insert** menu. This will show all three default planes for selection in a `Trimetric` orientation. A `Trimetric` orientation is a pictorial view that is oriented so the three mutually perpendicular planes appear unequally foreshortened.



From the screen, choose the `Front Plane`. The plane will highlight and rotate.

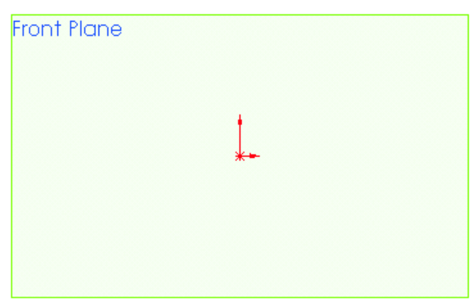
Note

The **Reference Triad** (lower left corner) shows the orientation of the model coordinate axes (red-`X`, green-`Y` and blue-`Z`) at all times. It can help show how the view orientation has been changed relative to the `Front Plane`.



4 Sketch active.

The selected `Front Plane` rotates so it is parallel to the screen. This only happens for the first sketch in a part.



The symbol represents the part's model origin which is the intersection of the X, Y, and Z axes. It is displayed in the color red, indicating that it is active.

Introducing: Confirmation Corner

When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

Sketch Indicator

When a sketch is active, or open, the confirmation corner displays two symbols. One looks like a sketch. The other is a red X. These symbols provide a visual reminder that you are active in a sketch. Clicking the sketch symbol exits the sketch and *saves any changes*. Clicking the red X exits the sketch and discards any changes.

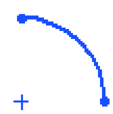
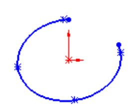
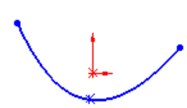
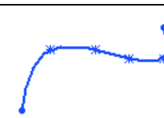
When other commands are active, the confirmation corner displays a check mark and an X. The check mark executes the current command. The X cancels the command.

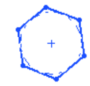
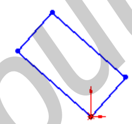
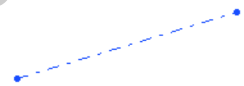
Sketch Entities

SolidWorks offers a rich variety of sketch tools for creating profile geometry. In this lesson, only one of the most basic shapes will be used: **Lines**.

Sketch Geometry

The following chart lists the basic sketch entities that are available by default on the Sketch toolbar.

Sketch Entity	Toolbar Button	Geometry Example
Line		
Circle		
Centerpoint Arc		
Tangent Arc		
3 Point Arc		
Ellipse		
Partial Ellipse		
Parabola		
Spline		

Sketch Entity	Toolbar Button	Geometry Example
Polygon		
Rectangle		
Parallelogram		
Point		
Centerline		

Basic Sketching

The best way to begin sketching is by using the most fundamental shape, the **Line**.

The Mechanics of Sketching

To sketch geometry, there are two techniques that can be used:

- **Click-Click**
  Position the cursor where you want the line to start. Click (press and release) the left mouse button. Move the cursor to where you want the line to end. A preview of the sketch entity will follow the cursor like a rubber band. Click the left mouse button a second time.

- **Click-Drag**
  Position the cursor where you want the line to start. Press and hold the left mouse button. Drag the cursor to where you want the sketch entity to end. A preview of the sketch entity will follow the cursor like a rubber band. Release the left mouse button.

Introducing: Insert Line

The **Line** tool creates single line segments in a sketch. Horizontal and vertical lines can be created while sketching by watching for the feedback symbols on the cursor.

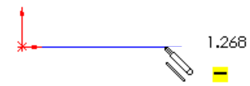
Where to Find It

- From the **Tools** menu, select **Sketch Entities, Line**.
- Or, with the cursor in the graphics window, right-click and select **Line** from the shortcut menu.
- Or, on the Sketch toolbar click **Line**.

Introducing: Sketch Relations

Sketch Relations are used to force a behavior on a sketch element thereby capturing design intent. They will be discussed in detail in *Sketch Relations* on page 33.

1 Sketch a line.

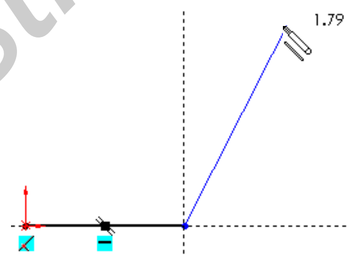


Click the **Line** tool and sketch a horizontal line from the origin. The “ – ” symbol appears at the cursor, indicating that a **Horizontal** relation is automatically added to the line. The number indicates the length of the line. Click again to end the line.

Important! Do not be too concerned with making the line the exact length. SolidWorks software is dimension driven – the dimensions control the size of the geometry, not the other way around. Make the sketch approximately the right size and shape and then use dimensions to make it exact.

2 Line at angle.

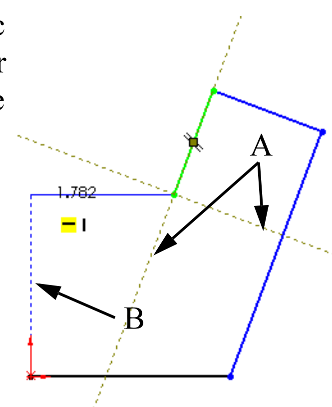
Starting at the end of the first line, sketch a line at an angle.



Inference Lines (Automatic Relations)

In addition to the “ – ” and “ | ” symbols, dashed inference lines will also appear to help you “line up” with existing geometry. These lines include existing line vectors, normals, horizontals, verticals, tangents and centers.

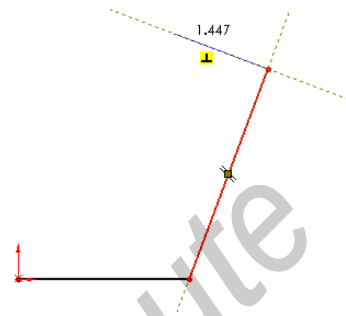
Note that some lines capture actual geometric relations, while others simply act as a guide or reference when sketching. A difference in the color of the inference lines will distinguish them. In the picture at the right, the lines labeled “A” are olive-green and if the sketch line snaps to them, will capture either a tangent or perpendicular relationship. The line labeled “B” is blue. It only provides a reference, in this case vertical, to the other endpoint. If the sketch line is ended at this point, no vertical relation will be captured.



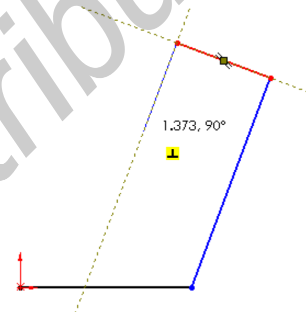
Note The display of Sketch Relations that appear automatically can be toggled on and off using **View, Sketch Relations**. They will remain on during the initial phase of sketching.

3 Inference lines.
Moving in a direction perpendicular to the previous line causes inference lines to be displayed. A **Perpendicular** relation is created between this line and the last one.

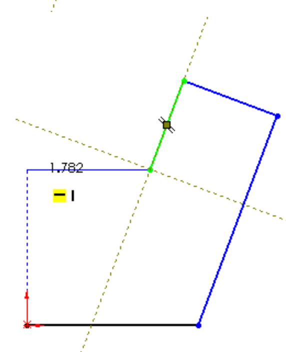
The cursor symbol indicates that you are capturing a perpendicular relation. Note that the line cursor is not shown for clarity.



4 Perpendicular.
Another perpendicular line is created from the last endpoint. Again, a perpendicular relation is automatically captured.

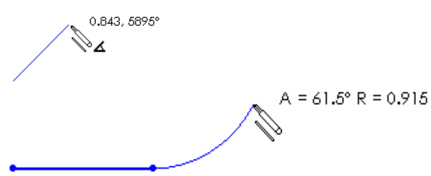


5 Reference.
Some inferences are strictly for reference and do *not* create relations. They are displayed in blue. This reference is used to align the endpoint vertically with the origin.

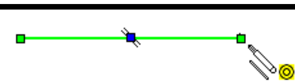


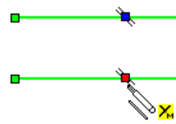
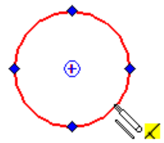
Sketch Feedback

The sketcher has many feedback features. The cursor will change to show what type of entity is being created. It will also indicate what selections on the existing geometry, such as end, coincident (on) or midpoint, are available using a red dot when the cursor is on it.

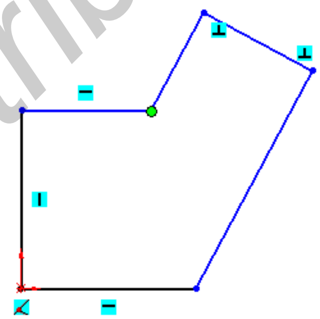


Three of the most common feedback symbols are:

Endpoint		Yellow concentric circles appear at the Endpoint when the cursor is over it.

Midpoint		The Midpoint appears as a square. It changes to red when the cursor is over the line.
Coincident (On Edge)		The quadrant points of the circle appear with a concentric circle over the centerpoint.

6 Close.
Close the sketch with a final line connected to the starting point of the first line.



Turning Off Tools

Turn off the active tool using *one* of these techniques:

- Press the **Esc** key on the keyboard.
- Click the **Line** tool a second time.
- Click the **Select** tool.
- Right-click in the graphics area, and choose **Select** from the shortcut menu.

Status of a Sketch

Sketches can be in one of three definition states at any time. The status of a sketch depends on geometric relations between geometry and the dimensions that define it. The three states are:

Under Defined

There is inadequate definition of the sketch, but the sketch can still be used to create features. This is good because many times in the early stages of the design process, there isn't sufficient information to fully define the sketch. When more information becomes available, the remaining definition can be added at a later time. Under defined sketch geometry is blue (by default).

Fully Defined

The sketch has complete information. Fully defined geometry is Black (by default). As a general rule, when a part is released to manufacturing, the sketches within it should be fully defined.

Over Defined

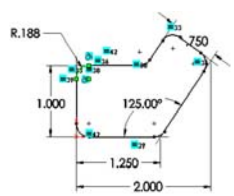
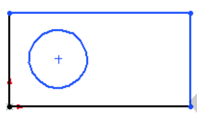
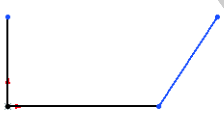
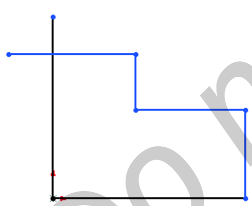
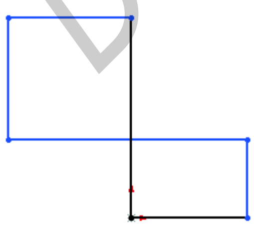
The sketch has duplicate dimensions or conflicting relations and it should not be used until repaired. Extraneous dimensions and relations should be deleted. Over defined geometry is red (by default).

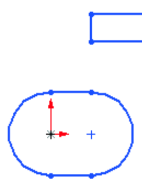
Additional Colors

There are several additional colors and states that may appear for geometry in the sketch. **Dangling** (brown), **Not Solved** (pink) and **Invalid** (yellow) all indicate errors that must be repaired.

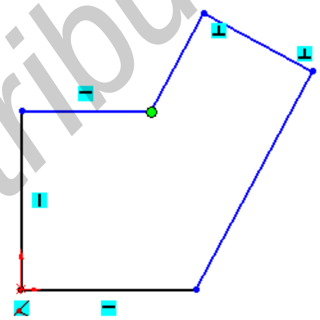
Rules That Govern Sketches

Different types of sketches will yield different results. Several different types are summarized in the table below. It is important to note that some of the techniques shown in the table below are advanced techniques that are covered either later in this course, or in other advanced courses.

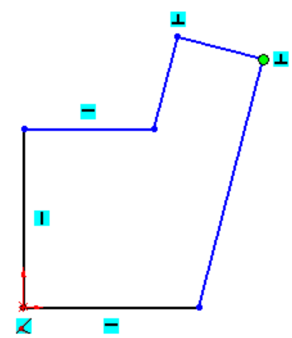
Sketch Type	Description	Special Considerations
	A typical "standard" sketch that is a neatly closed contour.	None required.
	Multiple nested contours creates a boss with an internal cut.	None required.
	Open contour creates a thin feature with constant thickness.	None required. For more information, see *Thin Features* on page 359.
	Corners are not neatly closed. *They should be.*	Use the **Contour Select Tool**. For more information, see *Sketch Contours* on page 258. Although this sketch will work, it represents poor technique and sloppy work habits. Do not do it.
	Sketch contains a self-intersecting contour.	Use the **Contour Select Tool**. For more information, see *Sketch Contours* on page 258. If both contours are selected, this type of sketch will create a **Multibody Solid**. See *Multibody Solids* in the *Advanced Part Modeling* course. Although this will work, multibodies are an advanced modeling technique that you should not use until you have more experience.

	The sketch *of the first feature* contains disjoint contours.	This type of sketch can create a **Multibody Solid**. See Multibody *Solids* in the *Advanced Part Modeling* course. Although this will work, multibodies are an advanced modeling technique that you should not use until you have more experience.

7 Current sketch status.
The sketch is **Under Defined** because some of the geometry is blue. Note that endpoints of a line can be a different color and different state than the line itself. For example, the vertical line at the origin is black because it is (a) vertical, and (b) attached to the origin. However, the uppermost endpoint is blue because the length of the line is under defined.



8 Dragging.
Under defined geometry (blue) can be dragged to new locations. Fully defined geometry cannot. Drag the uppermost endpoint to change the shape of the sketch. The dragged endpoint appears as a green dot.



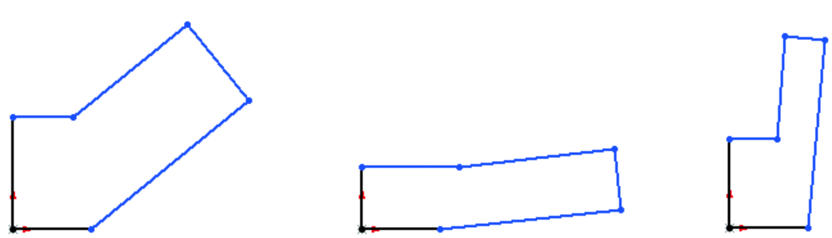
9 Undo the change.
Undo the last command by clicking the **Undo** option. You can see (and select from) a list of the last few commands by clicking the down arrow menu. The keyboard shortcut for **Undo** is **Ctrl+Z**.

Tip

You can also **Redo** a change, which reverts it back to the state prior to undo. The shortcut for redo is **Ctrl+Y**.

Design Intent

The design intent, as discussed earlier, governs how the part is built and how it will change. In this example, the sketch shape must be allowed to change in these ways:



What Controls Design Intent?

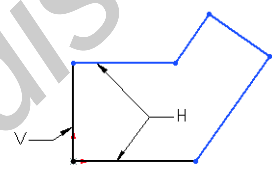
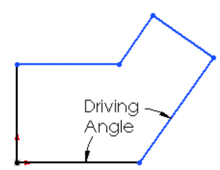
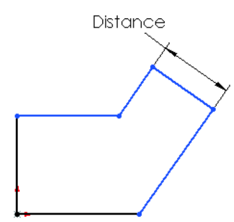
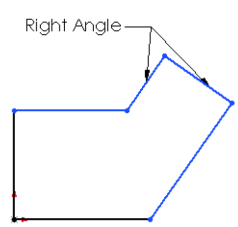
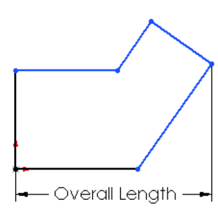
Design intent in a sketch is captured and controlled by a combination of two things:

- **Sketch relations**
  Create geometric relationships such as parallel, collinear, perpendicular, or coincident between sketch elements.
- **Dimensions**
  Dimensions are used to define the size and location of the sketch geometry. Linear, radial, diameter and angular dimensions can be added.

To fully define a sketch *and* capture the desired design intent requires understanding and applying a combination of relations and dimensions.

Desired Design Intent

In order for the sketch to change properly, the correct relations and dimensions are required. The required design intent is listed below:

Horizontal and vertical lines.	
Angle value.	
Parallel Distance value.	
Right-angle corners, or perpendicular lines.	
Overall length value.	

Sketch Relations

Sketch Relations are used to force a behavior on a sketch element thereby capturing design intent. Some are automatic, others can be added as needed. In this example, we will look at the relations on one of the lines and examine how they affect the design intent of the sketch.

Automatic Sketch Relations

Automatic relations are added as geometry is sketched. We saw this as we sketched the outline in the pervious steps. Sketch feedback tells you when automatic relations are being created.

Added Sketch Relations

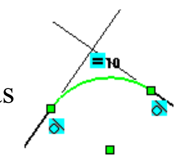
For those relations that cannot be added automatically, tools exist to create relations based on selected geometry and add dimensions.

Introducing: Display Relations

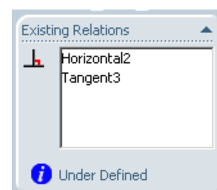
Display Relations shows and optionally allows you to remove geometric relationships between sketch elements.

Where to Find It

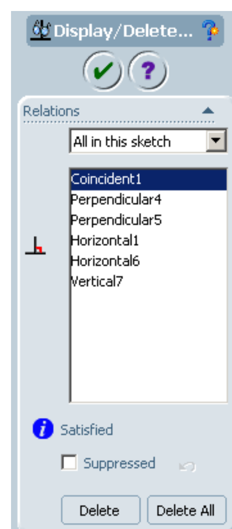
- Double-click the entity.
  Symbols appear indicating what relations are associated with that entity. In this example, the line has two relations: horizontal and tangent.



- The PropertyManager.
  Select the sketch entity and the PropertyManager shows the relations associated with that entity.

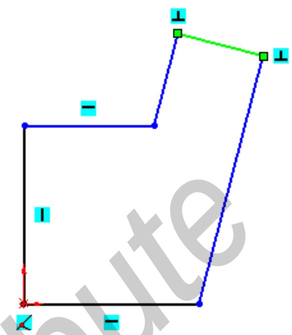


- Click **Display/Delete Relations** on the Dimensions/Relations toolbar.
  The PropertyManager will show a list of all the relations in the sketch.



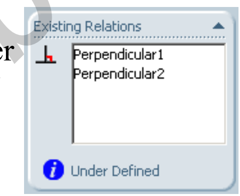
10 Display the relations associated with a line.

Double-click the uppermost angled line. Symbols appear identifying the lines that are perpendicular to the line you selected.



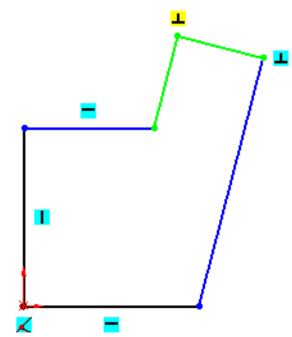
11 PropertyManager.

When you double-click the line, the PropertyManager opens. The **Relations** box in the PropertyManager also lists the geometric relations that are associated with the selected line.



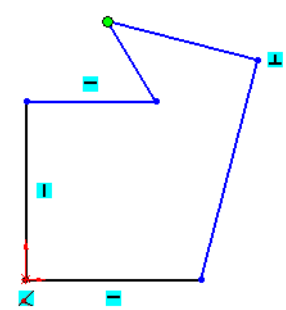
12 Remove the relation.

Remove the uppermost relation by clicking the relation, either the symbol or in the PropertyManager, and pressing the **Delete** key. If the symbol is selected, it turns yellow and displays the entitie(s) it controls.



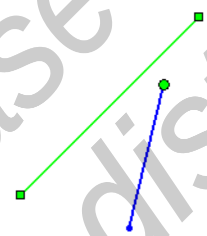
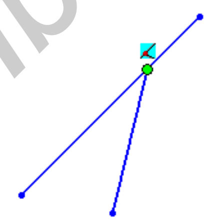
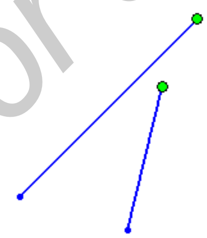
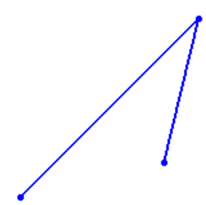
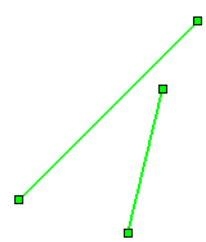
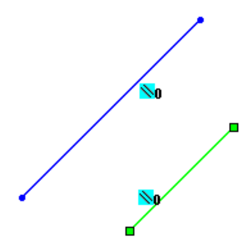
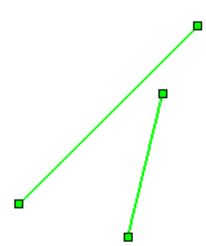
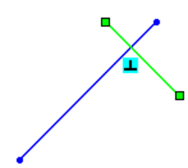
13 Drag the endpoint.

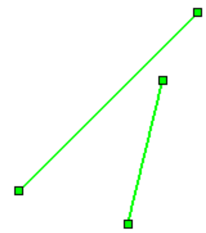
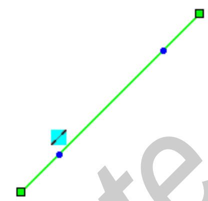
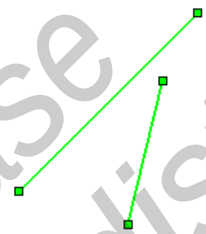
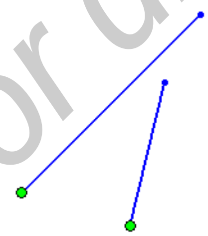
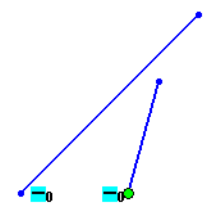
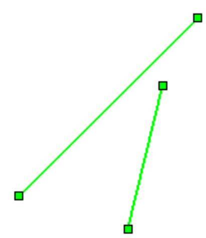
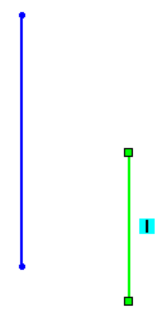
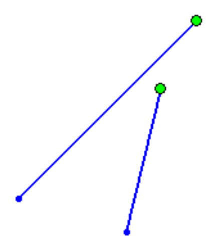
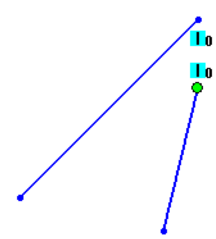
Because the line is no longer constrained to be perpendicular, the sketch will behave differently when you drag it. Compare this to how the sketch behaved when you dragged it in step **8**.

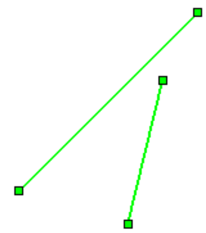
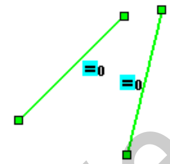
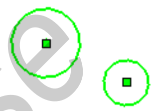
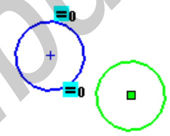
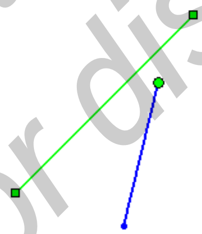
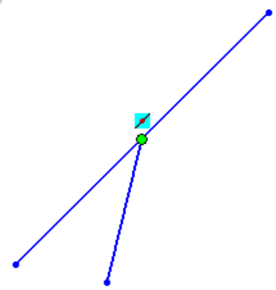


Examples of Sketch Relations

There are many types of **Sketch Relations**. Which ones are valid depends on the combination of geometry that you select. Selections can be the entity itself, endpoints or a combination. Depending on the selection, a limited set of options is made available. The following chart shows some examples of sketch relations. This is not a complete list of all geometric relations. Additional examples will be introduced throughout this course.

Relation	Before	After
Coincident between a line and an endpoint.		
Merge between two endpoints.		
Parallel between two lines.		
Perpendicular between two lines.		

Relation	Before	After
Collinear between two lines.		
Horizontal applied to one or more lines.		
Horizontal between two endpoints.		
Vertical applied to one or more lines.		
Vertical between two endpoints.		

Relation	Before	After
Equal between two lines.		
Equal between two arcs or circles		
Midpoint between a line and an endpoint.		

Introducing: Add Relations

Add Relations is used to create a geometric relationship such as parallel or collinear between sketch elements.

Where to Find It

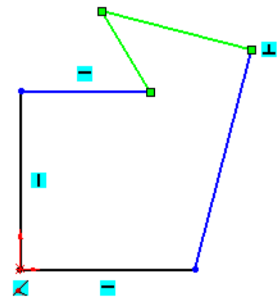
- Select the sketch entity or entities, and select the appropriate relation from the **Add Relations** section of the PropertyManager.
- Or, right-click the entity or entities, and select **Add Relation** from the short-cut menu.
- Or, click **Tools, Relations, Add...**
- Or, on the Sketch toolbar click **Add Relation** .

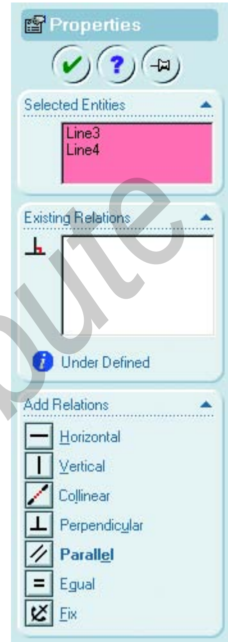
Selecting Multiple Objects

As you learned in Lesson 1, you select objects with the left mouse button. What about when you need to select more than one object at a time? When selecting multiple objects, SolidWorks follows standard Microsoft® Windows conventions: **Ctrl-select**. Hold down the **Ctrl** key while selecting the objects.

14 Add a relation.

Hold down **Ctrl** and select the two lines. The PropertyManager shows only those relations that are valid for the geometry selected.

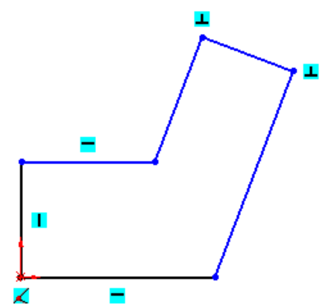




Click **Perpendicular**, and click **OK**.

15 Drag the sketch.

Drag the sketch back into approximately its original shape.



Dimensions

Dimensions are another way to define geometry and capture design intent in the SolidWorks system. The advantage of using a dimension is that it is used to both display the current value and change it.

Introducing: Smart Dimensions

The **Smart Dimension** tool determines the proper type of dimension based on the geometry chosen, *previewing* the dimension before creating it. For example, if you pick an arc the system will create a radial dimension. If you pick a circle, you will get a diameter dimension, while selecting two parallel lines will create a linear dimension between them. In cases where the **Smart Dimension** tool isn't quite smart enough, you have the option of selecting endpoints and moving the dimension to different measurement positions.

Where to Find It

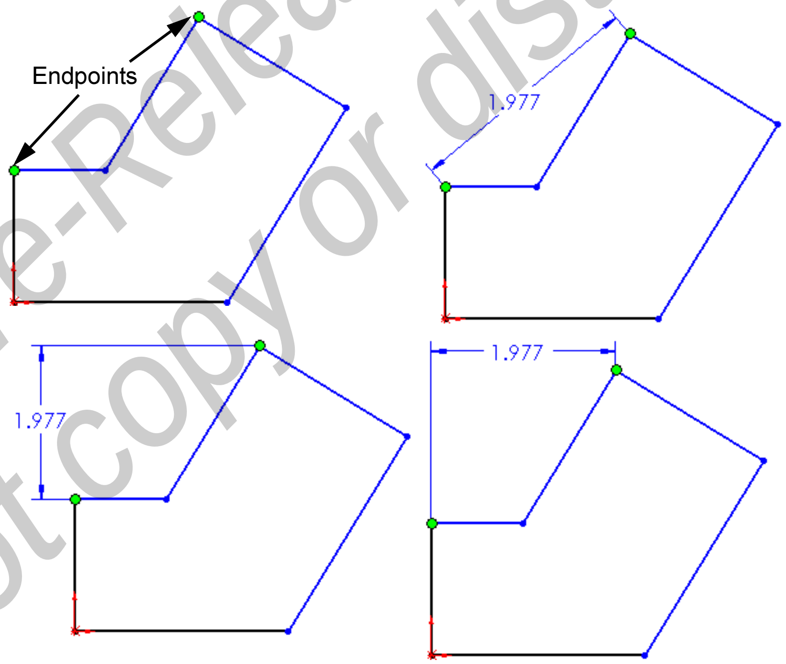
- From the **Tools** menu, select **Dimensions, Smart**.
- Or, right-click and select **Smart Dimension** from the shortcut menu.

- Or, on the Dimensions/Relations toolbar, pick the **Smart Dimension** tool.

Dimensioning: Selection and Preview

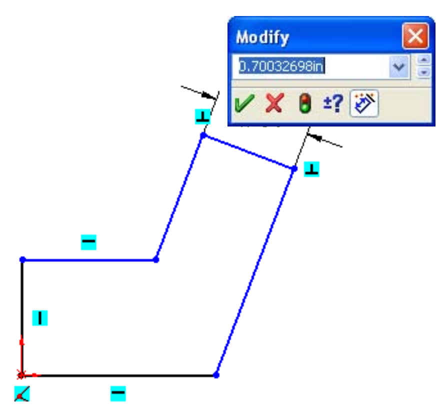
As you select sketch geometry with the dimension tool, the system creates a preview of the dimension. The preview allows you to see all the possible options by simply moving the mouse after making the selections. Clicking the left mouse button places the dimension in its current position and orientation. Clicking the right mouse button locks only the orientation, allowing you to move the text before final placement by clicking the left mouse button.

With the dimension tool and two endpoints selected, below are three possible orientations for a linear dimension. The value is derived from the initial point to point distance and may change based on the orientation selected.



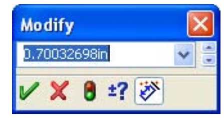
16 Adding a linear dimension.

Choose the dimension tool from any source and click the line shown. Click a second time to place the text of the dimension above and to the right of the line. The dimension appears with a **Modify** tool displaying the current length of the line. The spin box is used to incrementally increase/decrease the value. Or with the text highlighted, you can type a new value to change it directly.



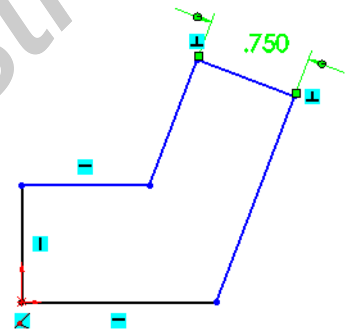
The Modify Tool

The modify tool that appears when you create or edit a dimension (parameter) has several options. The options available to you are:



- Spin the value up or down by a preset amount.
- Save the current value and exit the dialog box.
- Restore the original value and exit the dialog box.
- Rebuild the model with the current value.
- Change the spin increment value.
- Mark the dimension for drawing import.

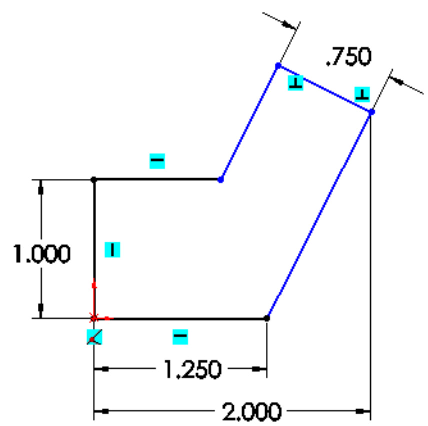
17 Set the value.
Change the value to **0.75** and click the **Save** option. The dimension forces the length of the line to be 0.75 inches.



Tip

Pressing **Enter** has the same effect as clicking the **Save** button.

18 Linear dimensions.
Add additional linear dimensions to the sketch as shown.



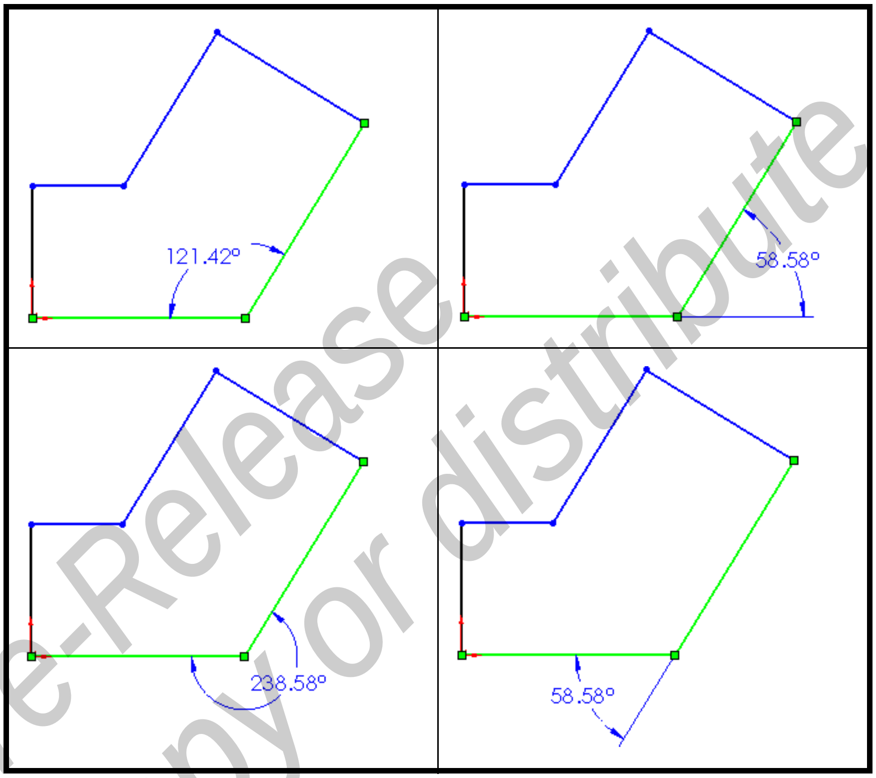
Dimensioning Tip

When you dimension a sketch, start with the smallest dimension first, and work your way to the largest.

Angular Dimensions

Angular dimensions can be created using the same dimension tool used to create linear, diameter and radial dimensions. Select either two lines that are both non-collinear and non-parallel, or select three non-collinear endpoints.

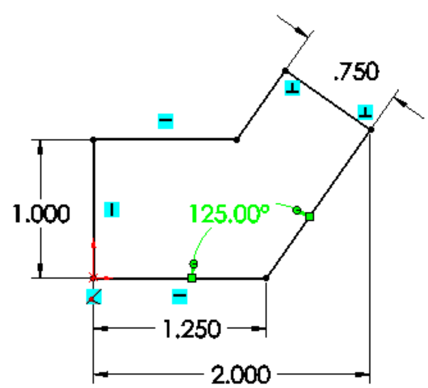
Depending on where you place the angular dimension, you can get the interior or exterior angle, the acute angle, or the oblique angle. Possible placement options:



19 Angular dimension.

Using the dimension tool, create the angular dimension shown and set the value to **125°**.

The sketch is fully defined.



Sketch Fillets

Sketch Fillets are used to round off sharp corners in a sketch. A sketch fillet can be applied to a sketch that is already fully defined.

Important!

Not all fillets should be added at the sketch level. There is a fillet command that works directly on solid models that may be more appropriate to use. You will learn about this in later lessons.

Introducing: Sketch Fillet

Sketch Fillet is used to create a fillet or round in a sketch. The fillet is created as an arc placed tangent to adjacent entities.

Where to Find It

- From the **Tools** menu, select **Sketch Tools, Fillet**.
- Or, on the Sketch toolbar click **Sketch Fillet**.

Note

Sketch Fillets do not allow **0** radius values.

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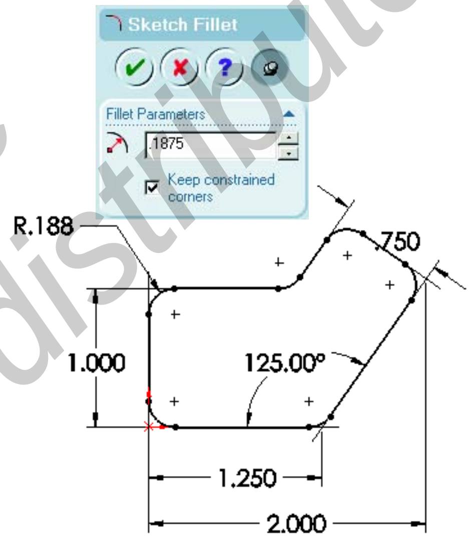
20 Sketch fillets.

Click **Sketch Fillet** and set the **Radius** to **0.1875”**. Select all of the endpoints in the sketch.

Click **OK**.

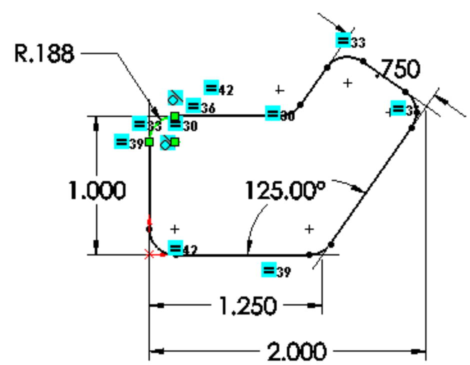


Note

For clarity, the relations are hidden in this example and the remainder of the lesson.

Why is There Only One Dimension?

A dimension only appears on the first fillet you create. All the fillets created in one operation are controlled by this dimension value. How is this done? Displaying the relations on the first fillet shows the answer: the system automatically adds an **Equal** relation to the other fillets in the series.



Extrude

Once the sketch is completed, it can be extruded to create the first feature. There are many options for extruding a sketch including the end conditions, draft and depth of extrusion, which will be discussed in more detail in later lessons. Extrusions take place in a direction normal to the sketch plane, in this case the `Front` plane.

Where to Find It

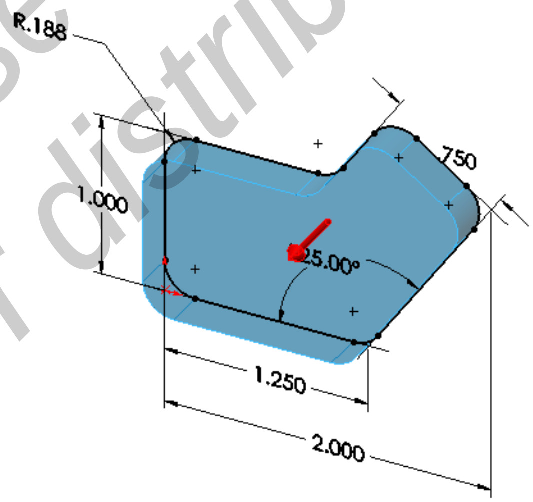
- From the menu: **Insert, Boss/Base, Extrude...**.
- Or, on the Features toolbar, choose: .

21 Extrude menu.

Click **Insert, Boss/Base, Extrude** or the tool on the Features toolbar to access the command.

On the **Insert** menu, the options for other methods of creating features are listed along with **Extrude** and **Revolve**. They are unavailable because this sketch does not meet the conditions necessary for creating these types of features. For example, a **Sweep** feature requires both profile and path sketches. Since there is only one sketch at this time, the **Sweep** option is unavailable.

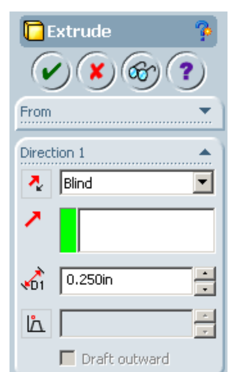
22 Preview graphics.
The view orientation automatically changes to `Trimetric` and a preview of the feature is shown at the default depth.



Handles appear that can be used to drag the preview to the desired depth. The handles are colored red for the active direction and gray for inactive direction. A callout shows the current depth value.

Tip Color settings in SolidWorks can be modified using **Tools, Options**.

23 Extrude Feature settings.
Change the settings as shown.



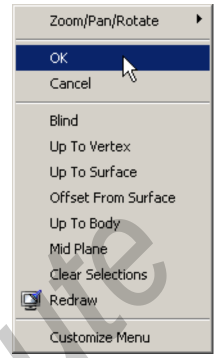
- End Condition = **Blind**
- (Depth) = **0.25"**

Click **OK** to create the feature.

Tip The **OK** button is just one way to accept and complete the process.

A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.

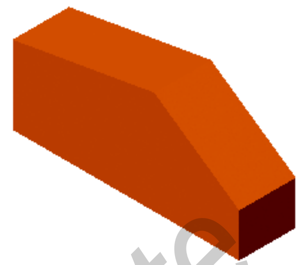
A third method is to right-click and select **OK** from the shortcut menu.



24 Completed feature.
The completed feature is the first solid, or feature of the part. The sketch is absorbed into the `Extrude1` feature.

25 Save and close.
Click **Save** to save your work and click **Close** to close the part.

Exercise 1: Sketching Horizontal and Vertical Lines



Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

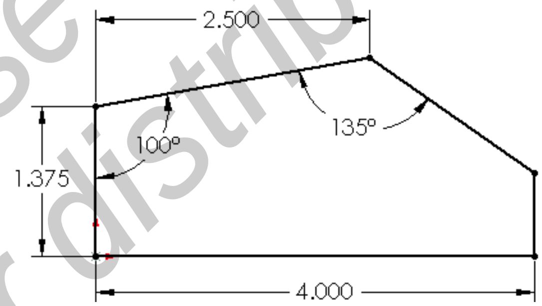
This lab reinforces the following skills:

- Sketching.
- Dimensions.
- Extruding a feature.

1 **New part.**
Open a new part using the `Part_IN` template.

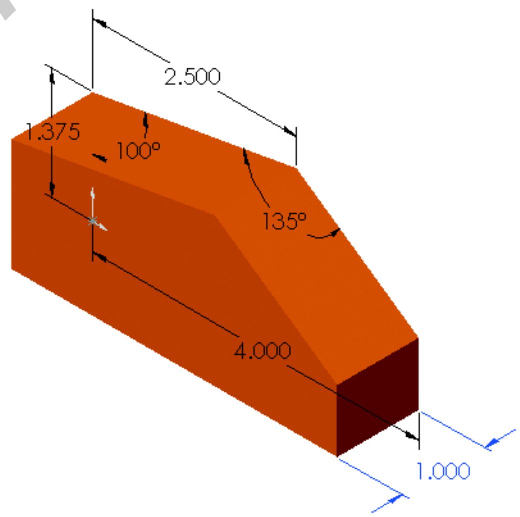
2 **Sketch.**
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions.

Fully define the sketch.



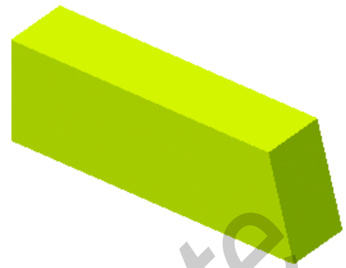
3 **Extrude.**
Extrude the sketch **1”** in depth.

4 **Save and close the part.**



Exercise 2: Sketching Lines with Inferences

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.



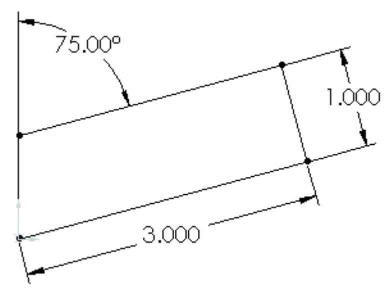
This lab reinforces the following skills:

- Sketching.
- Dimensions.
- Extruding a feature.

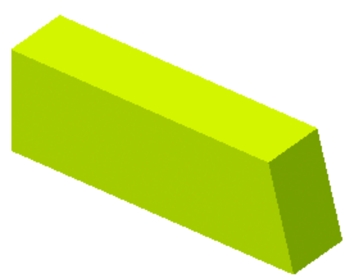
1 New part.
Open a new part using the `Part_IN` template.

2 Automatic relations.
Create this sketch on the `Front Plane` using lines and automatic relations. Show the **Perpendicular** and **Vertical** relations.

3 Dimensions.
Add dimensions to fully define the sketch.



4 Extrude.
Extrude the sketch **0.5”**.



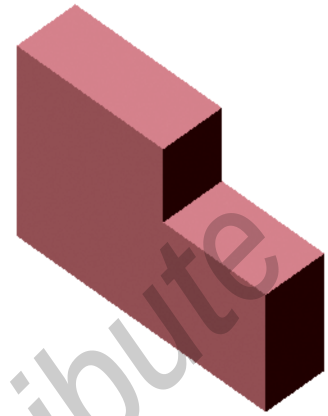
5 Save and close the part.

Exercise 3: Sketching Lines

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

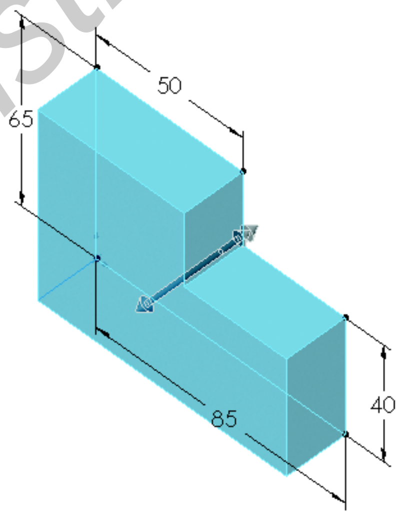
This lab reinforces the following skills:

- Sketching.
- Dimensions.
- Extruding a feature.

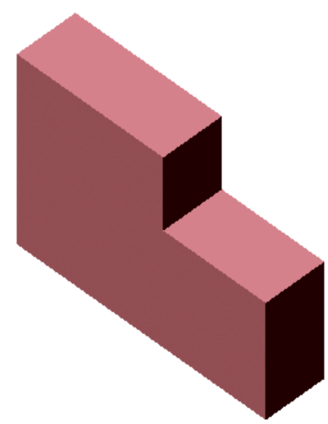


1 New part.
Open a new part using the `Part_MM` template.

2 Sketch and extrude.
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions. Extrude the sketch **20mm** in depth.



3 Save and close the part.



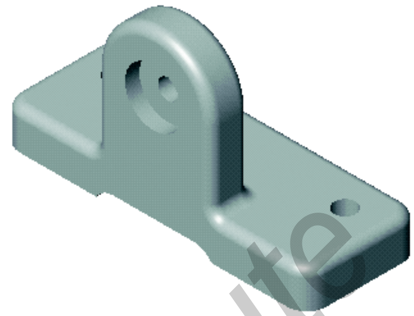
Lesson 3
Basic Part Modeling

Upon successful completion of this lesson, you will be able to:

- Choose the best profile for sketching.
- Choose the proper sketch plane.
- Create a new part.
- Create a sketch.
- Extrude a sketch as a boss.
- Extrude a sketch as a cut.
- Create Hole Wizard holes.
- Insert fillets on a solid.
- Make a basic drawing of a part.
- Make a change to a dimension.
- Demonstrate the associativity between the model and its drawings.

Basic Modeling

This lesson discusses the considerations that you make before creating a part, and shows the process of creating a simple one.



Stages in the Process

The steps in planning and executing the creation of this part are listed below.

- **Terminology**
  What are the terms commonly used when talking about modeling and using the SolidWorks software?
- **Profile choice**
  Which profile is the best one to choose when starting the modeling process?
- **Sketch plane choice**
  Once you've chosen the best profile, how does this affect your choice of sketch plane?
- **Design intent**
  What is design intent and how does it affect the modeling process?
- **New part**
  Opening the new part is the first step.
- **First feature**
  What is the first feature?
- **Boss and hole features**
  How do you modify the first feature by adding bosses and holes?
- **Fillets**
  Rounding off the sharp corners – filleting.
- **Dimension changes**
  Making a change to a dimension changes the model's geometry. How does this happen?

Terminology

Moving to 3D requires some new terminology. The SolidWorks software employs many terms that you will become familiar with through using the product. Many are terms that you will recognize from design and manufacturing such as cuts and bosses.

Feature

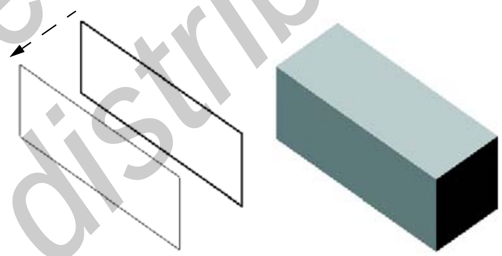
All cuts, bosses, planes and sketches that you create are considered Features. Sketched features are those based on sketches (boss and cut), applied features are based on edges or faces (fillet).

Plane

Planes are flat and infinite. They are represented on the screen with visible edges. They are used as the primary sketch surface for creating boss and cut features.

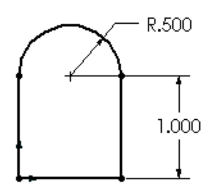
Extrusion

Although there are many ways to create features and shape the solid, for this lesson, only *extrusions* will be discussed. An extrusion will extend a profile along a path normal to the profile plane for some distance. The movement along that path becomes the solid model.



Sketch

In the SolidWorks system, the name used to describe a 2D profile is *sketch*. Sketches are created on flat faces and planes within the model. They are generally used as the basis for bosses and cuts, although they can exist independently.



Boss

Bosses are used to *add* material to the model. The critical initial feature is always a boss. After the first feature, you may add as many bosses as needed to complete the design. As with the base, all bosses begin with a sketch.

Cut

A *Cut* is used to *remove* material from the model. This is the opposite of the boss. Like the boss, cuts begin as 2D sketches and remove material by extrusion, revolution, or other methods you will learn about.

Fillets and Rounds

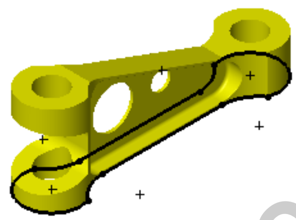
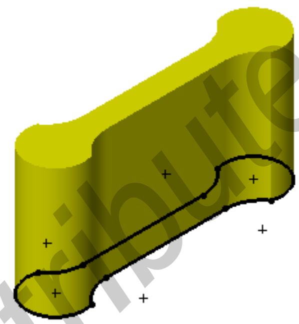
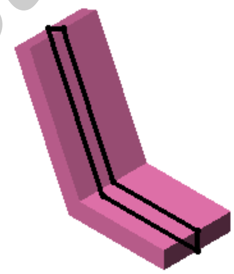
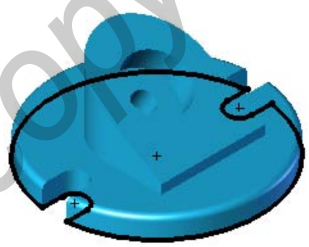
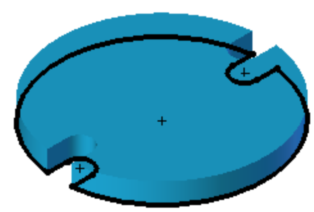
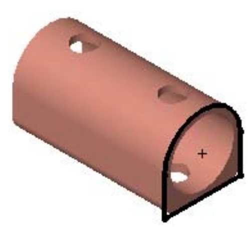
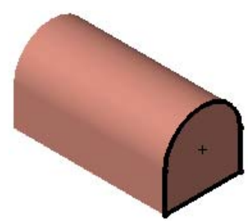
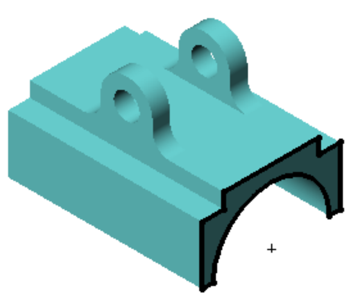
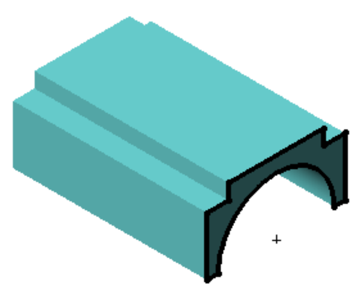
Fillets and *rounds* are generally added to the solid, not the sketch. By nature of the faces adjacent to the selected edge, the system knows whether to create a round (removing material) or a fillet (adding material).

Design Intent

How the model should be created and changed, is considered the **Design Intent**. Relationships between features and the sequence of their creation all contribute to design intent.

Choosing the Best Profile

Choose the *"best"* profile. This profile, when extruded, will generate more of the model than any other. Look at these models as examples.

Part	Best Profile Extruded
	
	
	
	
	

Choosing the Sketch Plane

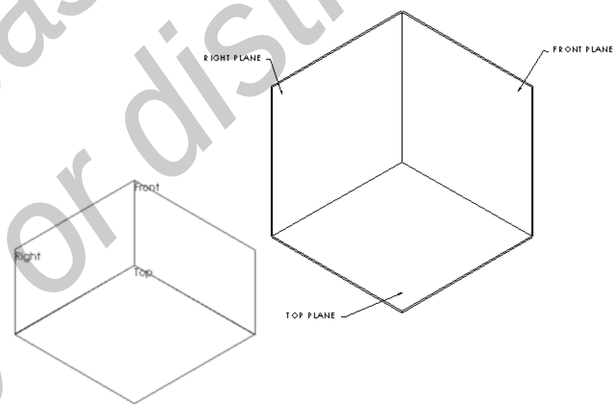
Once the best profile is determined, the next step is to decide which view to use and select the plane with the same name for sketching it. The SolidWorks software provides three reference planes, they are described below.

Reference Planes

There are three default reference planes, labeled `Front Plane`, `Top Plane` and `Right Plane`. Each plane is infinite, but has screen borders for viewing and selection. Also, each plane passes through the origin and is mutually perpendicular to the others.

The planes can be renamed. In this course the names `Front`, `Top` and `Right` replace the default names respectively. This naming convention is used in other CAD systems and is comfortable to many users.

Although the planes are infinite, it may be easier to think of them as forming an open box, connecting at origin. Using this analogy, the inner faces of the box are the potential sketch planes.



Placement of the Model

The part will be placed into the box three times. Each time the best profile will contact or be parallel to one of the three planes. Although there are many combinations, the choices are limited to three for this exercise.

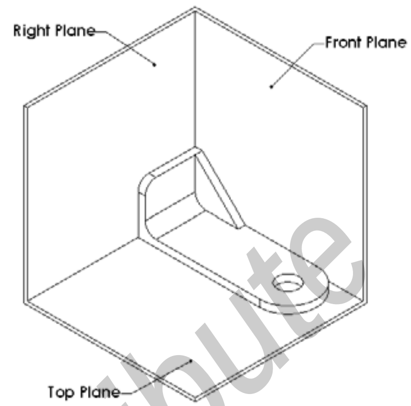
There are several things to consider when choosing the sketch plane. Two are appearance and the part's orientation in an assembly. The appearance dictates how the part will be oriented in standard views such as the `Isometric`. This also determines how you will spend most of your time looking at the model as you create it.

The part's orientation in an assembly dictates how it is to be positioned with respect to other, mating parts.

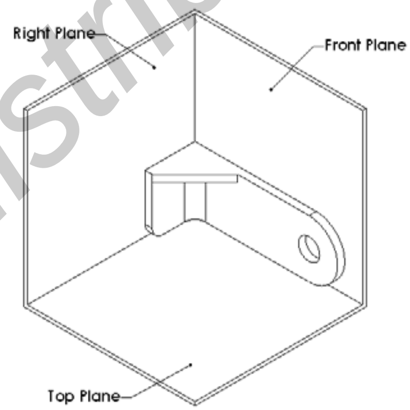
Orient the Model for the Drawing

Another consideration when deciding which sketch plane to use is how you want the model to appear on the drawing when you detail it. You should build the model so that the Front view is the same as the Front view will be in the final drawing. This saves time during the detailing process because you can use predefined views.

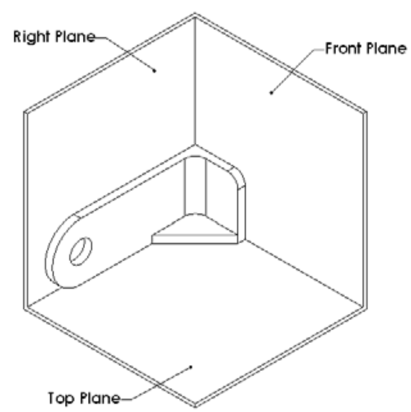
In the first example, the best profile is in contact with the `Top` plane.



In the second example, it is contacting the `Front` plane.



The last example shows the best profile in contact with the `Right` plane.

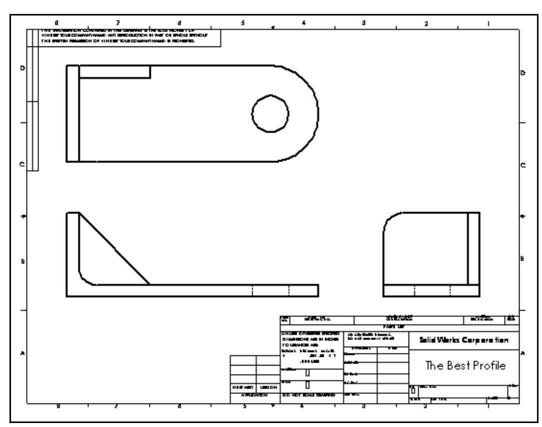


Chosen Plane

The `Top` plane orientation seems to be the best. This indicates that the best profile should be sketched on the `Top` plane of the model.

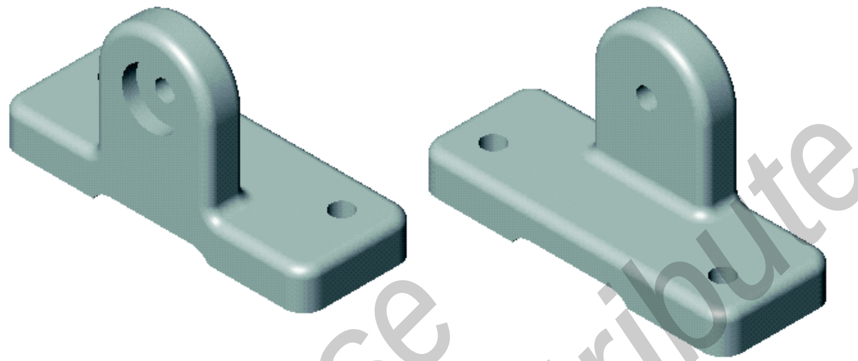
How it Looks on the Drawing

By giving careful thought to which plane is used to sketch the profile, the proper views are easily generated on the detail drawing.



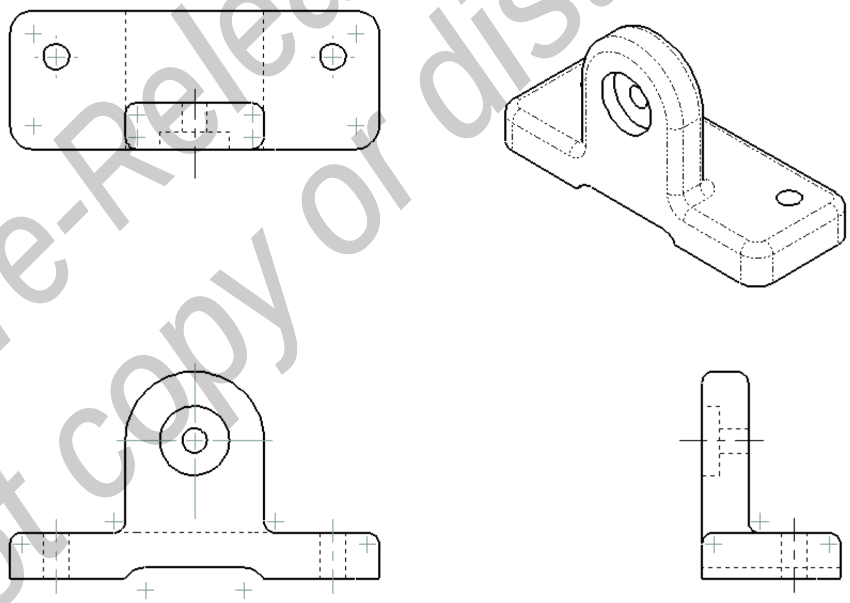
Details of the Part

The part we will be creating is shown at right. There are two main boss features, some cuts, and fillets.



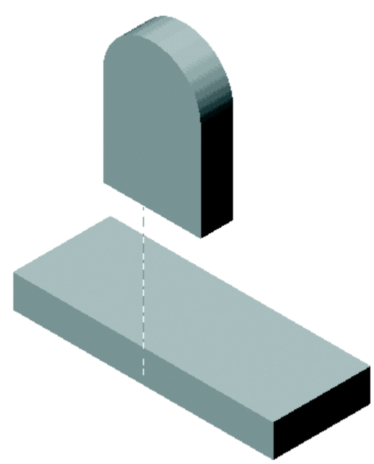
Standard Views

The part is shown here in four standard views.



Main Bosses

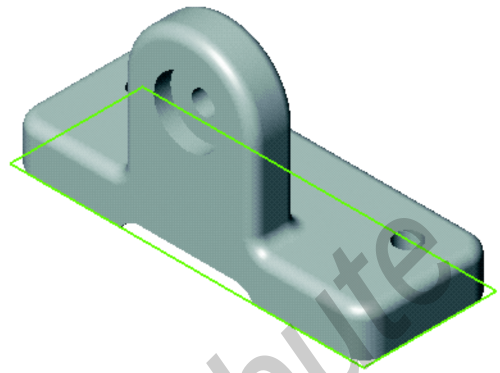
The two main bosses have distinct profiles in different planes. They are connected as shown in the exploded view at right.



Best Profile

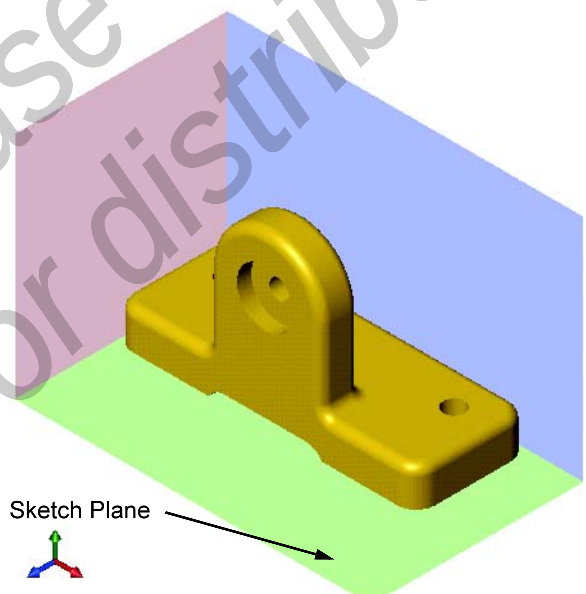
The first feature of the model, is created from the rectangular sketch shown overlaid on the model. This is the best profile to begin the model.

The rectangle will then be extruded as a boss to create the solid feature.



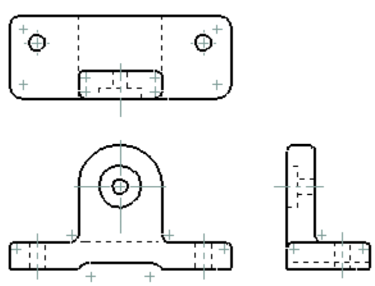
Sketch Plane

Placing the model "in the box" determines which plane should be used to sketch on. In this case it will be the `Top` reference plane.



Design Intent

The design intent of this part describes how the part's relationships should or should not be created. As changes to the model are made, the model will behave as intended.



- All holes are through holes.
- Holes in base are symmetrical.
- Slot is aligned with tab.

Procedure

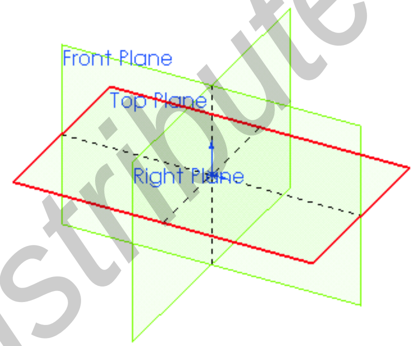
The modeling process includes sketching and creating bosses, cuts and fillets. To begin with, a new part file is created.

1 New part.

Click **New**, or click **File, New**. Create a new part using the `Part_IN` template and **Save** it as `Basic`.

2 Select the sketch plane.

Insert a new sketch and choose the `Top Plane`.



Tip

A plane doesn't have to be shown in order to be used; it can be selected from the FeatureManager.

Sketching the First Feature

Create the first feature by extruding a sketch into a boss. Begin with the sketch geometry, a rectangle.

Introducing: Insert Rectangle

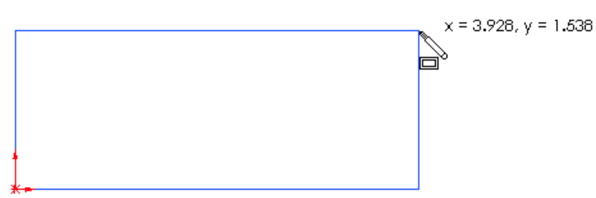
Insert Rectangle is used to create a rectangle in a sketch. The rectangle is comprised of four lines (two horizontal and two vertical) connected at the corners. It is sketched by indicating the locations of two diagonal corners.

Where to Find It

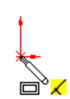
- On the Sketch toolbar, click **Rectangle**.
- Or, on the **Tools** menu, select **Sketch Entities, Rectangle**.

3 Sketch a rectangle.

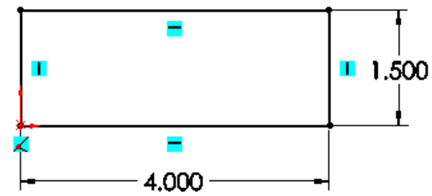
Click the **Rectangle** tool and begin the rectangle at the origin.



Make sure the rectangle is locked to the origin by looking for the *vertex* cursor as you begin sketching. Do not worry about the size of the rectangle. Dimensioning it will take care of that in the next step.



4 **Fully defined sketch.**
Add dimensions to the sketch. The sketch is fully defined.

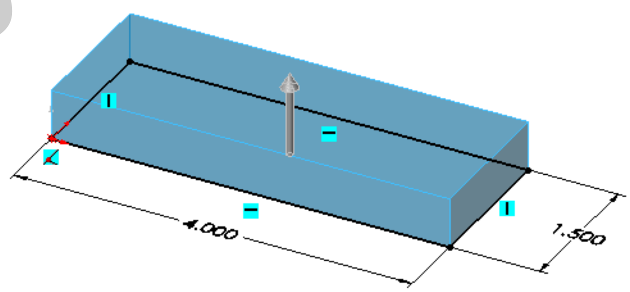


Extrude Options

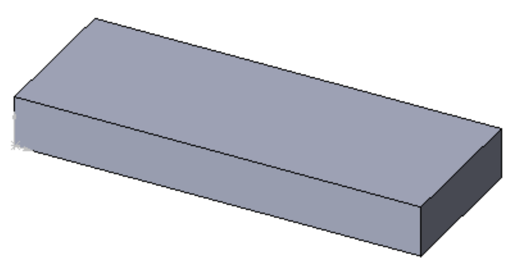
An explanation of some of the more frequently used **Extrude** options is given below. Other options will be discussed in later lessons.

- **End Condition Type**
  A sketch can be extruded in one or two directions. Either or both directions can terminate at some blind depth, up to some geometry in the model, or extend through the whole model.
- **Depth**
  The distance for a blind or mid-plane extrusion. For mid-plane, it refers to the total depth of the extrusion. That would mean that a depth of 50mm for a mid-plane extrusion would result in 25mm on each side of the sketch plane.
- **Draft**
  Applies draft to the extrusion. Draft on the extrusion can be inwards (the profile gets smaller as it extrudes) or outward.

5 **Extrude.**
Extrude the rectangle **0.5”** upwards.



The completed feature is shown at the right.



Renaming Features

Any feature that appears in the FeatureManager design tree (aside from the part itself) can be renamed. Renaming features is a useful technique for finding and editing features in later stages of the model. Well chosen, logical names help you to organize your work and make it easier when someone else has to edit or modify your model.

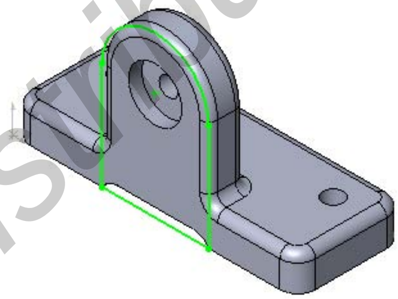
6 Rename the feature.
It is good practice to rename the features that you create with some meaningful name. In the FeatureManager design tree, use a very slow double-click to edit the feature `Extrude1`. When the name is highlighted and editable, type `BasePlate` as the new feature name. All features in the SolidWorks system can be edited in the same way.

Tip

Instead of using a slow double-click to edit the name, you can select the name and press **F2**.

Boss Feature

The next feature will be the boss with a curved top. The sketch plane for this feature is not an existing reference plane, but a planar face of the model. The required sketch geometry is shown overlaid on the finished model.



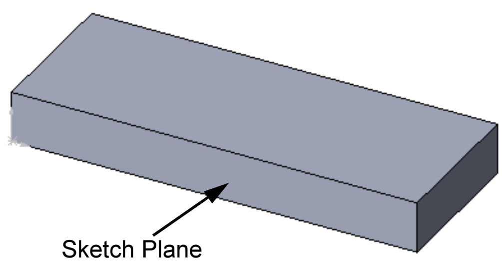
Tip

Cut features are created in the same way as bosses – with a sketch and extrusion. They remove material rather than add it.

Sketching on a Planar Face

Any planar (flat) face of the model can be used as a sketch plane. Simply select the face and choose the **Sketch** tool. Where faces are difficult to select because they are on the rear of the model or are obscured by other faces, the **Select Other** tool can be used to choose a face without reorienting the view. In this case, the planar face on the front of the `BasePlate` is used.

7 Insert new sketch.
Create a new sketch using **Insert, Sketch** or by clicking the **Sketch** tool . Select the indicated face.



Sketching

SolidWorks offers a rich variety of sketch tools for creating profile geometry. In this example **Tangent Arc** is used to create an arc that begins tangent to a selected endpoint on the sketch. Its other endpoint can be placed in space or on another sketch entity.

Introducing: Insert Tangent Arc

Insert Tangent Arc is used to create tangent arcs in a sketch. The arc must be tangent to some other entity, line or arc, at its start.

Where to Find It

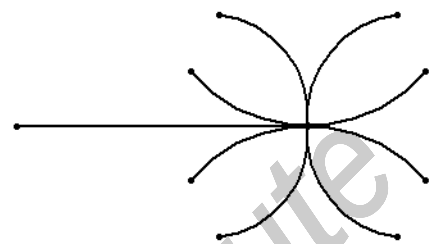
- From the **Tools** menu, select **Sketch Entities, Tangent Arc**.
- Or, with the cursor in the graphics window, right-click and select

Tangent Arc.

- Or, on the Sketch toolbar click **Tangent Arc**.

Tangent Arc Intent Zones

When you sketch a tangent arc, the SolidWorks software infers from the motion of the cursor whether you want a tangent or normal arc. There are four intent zones, with eight possible results as shown.



You can start sketching a tangent arc from the end point of any existing sketch entity (line, arc, spline, and so on). Move the cursor away from the end point.

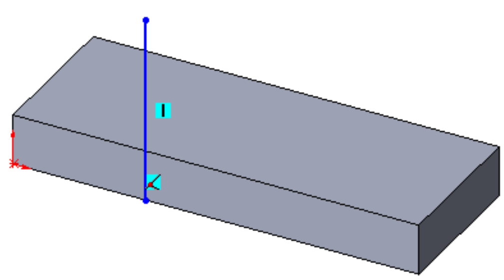
- Moving the cursor in a tangent direction creates one of the four tangent arc possibilities.
- Moving the cursor in a normal direction creates on of the four normal arc possibilities.
- A preview shows what type of arc you are sketching.
- You can change from one to the other by returning the cursor to the endpoint and moving away in a different direction.

Autotransitioning Between Lines and Arcs

When using the **Line** tool, you can switch from sketching a line to sketching a tangent arc, and back again, without selecting the **Tangent Arc** tool. You can do this by moving the cursor as described above, or by pressing the **A** key on the keyboard.

8 **Vertical line.**

Click the line tool and start the vertical line at the lower edge capturing a **Coincident** relation at the lower edge and **Vertical** relation .

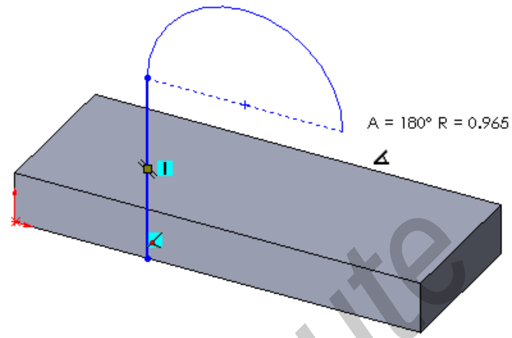


9 **Autotransition.**

Press the letter **A** on the keyboard.

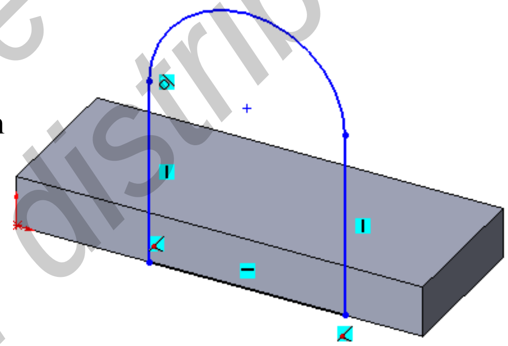
You are now in tangent arc mode.

10 Tangent arc.
Sketch a 180° arc tangent to the vertical line. Look for the inference line indicating that the end point of the arc is aligned horizontally with the arc's center.



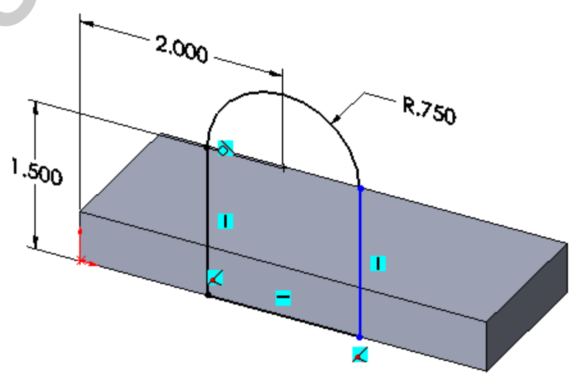
When you finish sketching the tangent arc, the sketch tool automatically switches back to the line tool.

11 Finishing lines.
Create a vertical line from the arc end to the base, and one more line connecting the bottom ends of the two vertical lines.



Note that the horizontal line is black, but its endpoints are not.

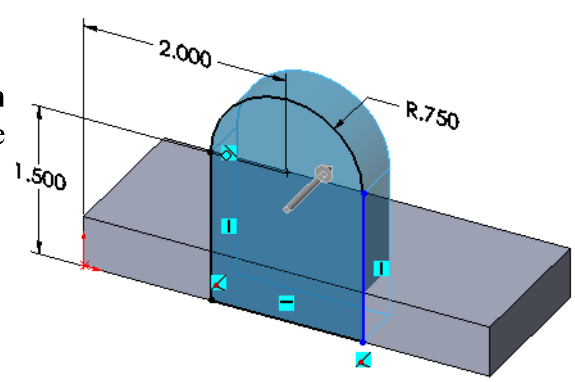
12 Add dimensions.
Add linear and radial dimensions to the sketch.



As you add the dimensions, move the cursor around to view different possible orientations.

Always dimension to an arc by selecting on its circumference, rather than center. This makes other dimensioning options (min and max) available.

13 Extrude direction.
Click **Insert, Boss, Extrude** and set the **Depth** to **0.5** inches. Note that the preview shows the extrusion going into the base, in the proper direction.

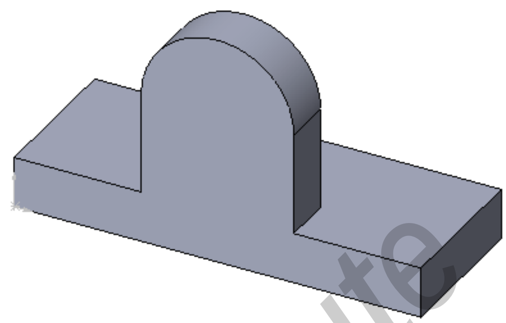


If the direction of the preview is away from the base, click the **Reverse direction** button.

14 Completed boss.

The boss merges with the previous base to form a single solid.

Rename the feature `VertBoss`.



Viewports

Viewports can be used to view and edit a model in multiple view orientations at the same time. The viewport icons include: **Single View**, **Two View** (horizontal and vertical), **Four View** and **Link Views**. **Link Views** ties the views together for zooming and panning.

Each viewport contains a view pop-up menu in the lower left corner. This menu displays the current view orientation (`Custom` for anything that is not a standard view) and contains a menu to change the view orientation.

Note

The default four viewport orientation, first angle or third angle, is set using **Tools, Options, System Options, Display/Selection, Projection type for four view viewport**.

Where to Find It

- From the Standard Views toolbar click the appropriate icon.
- Or, click the view pop-up menu and a select an icon.

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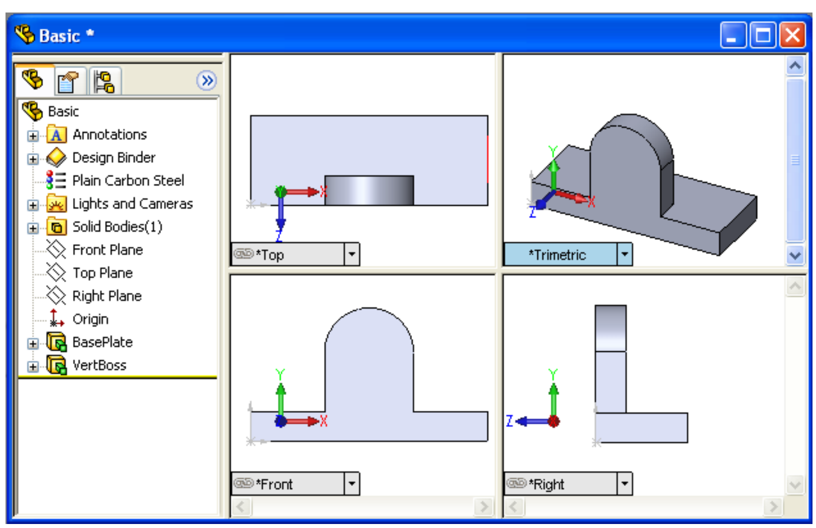
15 Four viewports.

Click **Four View** to divide the graphics window into four equal sized viewports. The view pop-up menu is blue in the active viewport.



Using the Hole Wizard

The **Hole Wizard** is used to create specialized holes in a solid. It can create simple, tapered, counterbored and countersunk holes using a step by step procedure. In this example, the **Hole Wizard** will be used to create a standard hole.

Creating a Standard Hole

You can choose the face to insert the hole onto, define the hole's dimensions and locate the hole and using the **Hole Wizard**. One of the most intuitive aspects of the **Hole Wizard** is that you specify the size of the hole by the fastener that goes into it.

Tip

You can also place holes on reference planes and non-planar faces. For example, you can create a hole on a cylindrical face.

Introducing: The Hole Wizard

The **Hole Wizard** creates shaped holes, such as countersunk and counterbore types. The process creates two sketches. One defines the shape of the hole. The other, a point, locates the center.

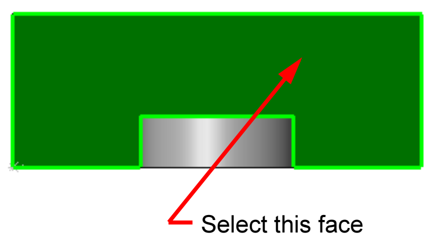
Note

The Hole Wizard requires a face to be selected or pre-selected, not a sketch.

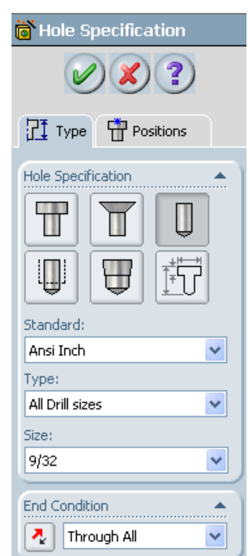
Where to Find It

- From the **Insert** menu choose **Features, Hole, Wizard...**.
- Or choose the **Hole Wizard** tool on the Features toolbar.

16 Select face.
Select the top, flat face of the base feature and click .



17 Type.
The **Hole Specification** dialog appears. Set the properties of the hole as follows:



Type: Hole

Standard: Ansi Inch

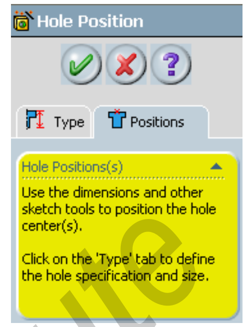
Screw Type: All Drill sizes

Size: 9/32

End Condition: Through All

Click the **Positions** tab.

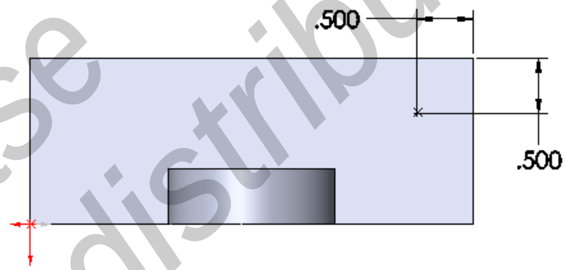
18 Positions.
A point and hole preview is placed on the selected face near where you selected it.



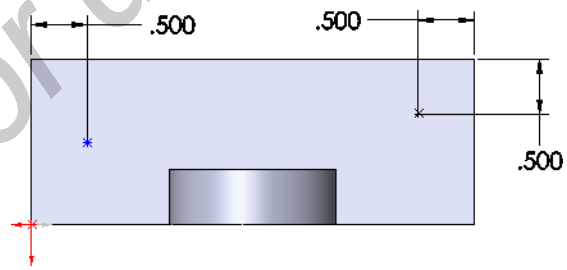
Tip

Multiple instances of the hole can be created in one command by inserting additional points at other locations.

19 Dimensions.
Add dimensions between the model edges and the point as shown.

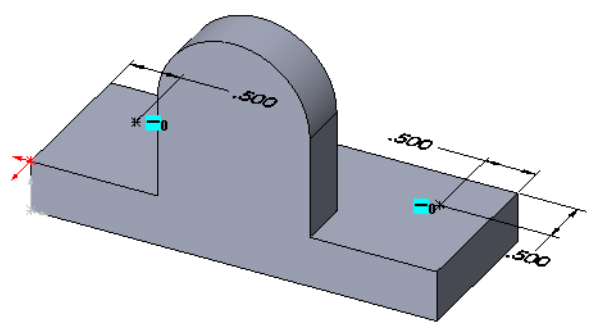


20 Additional point.
Click to add another Point on the face. Add the dimension shown.

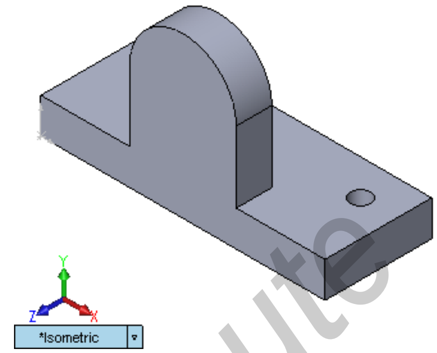


21 Horizontal Relation.
Press **Esc** to turn off the dimension tool. Select both points and add a **Horizontal** relation between them.

22 Single viewport.
Click **Single View** to return to the original view and click **OK**.



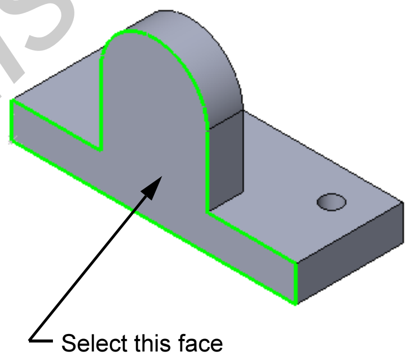
23 Change the view orientation.
Click the view orientation menu and choose `Isometric` to change view orientation.



Counterbore Hole

A counterbore hole is required in this model. Using the front face of the model and a relation, the hole can be positioned.

24 Hole position.
Again, an existing face of the model will be used to position geometry. Select the face indicated and **Insert, Features, Hole, Wizard...**.



25 Click Counterbore.
Set the properties of the hole as follows:

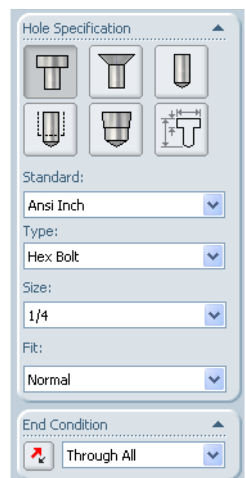
Standard: Ansi Inch

Screw Type: Hex Bolt

Size: 1/4

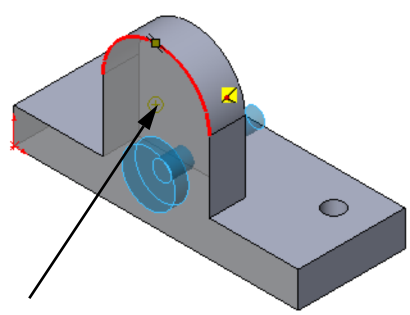
End Condition: Through All

Click the **Positions** tab.



26 Wake up the centerpoint.
Turn off the **Point** tool. Drag the point onto the circumference of the large arc. *Do not drop it.*

When the **Coincident** symbol appears, the center point of the large arc has been "woken up" and is now a point you can snap to.



Drop the point onto the arc's centerpoint. Look for the feedback that tells you that you are snapping to the arc's center, a coincident relation. Click **OK**.

Cut Feature

Once the two main boss features are completed, it is time to create a cut to represent the removal of material. Cut features are created in the same way as bosses- in this case with a sketch and extrusion.

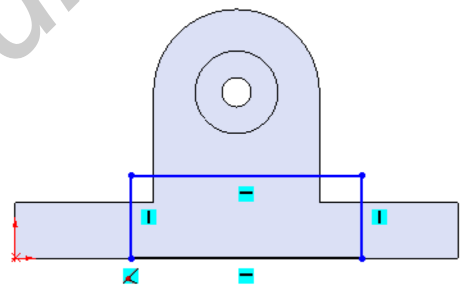
Introducing: Cut Extrude

The menu for creating a cut feature by extruding is identical to that of creating a boss. The only difference is that a cut removes material while a boss adds it. Other than that distinction, the commands are the same. this cut represents a slot.

Where to Find It

- From the **Insert** menu, select **Cut, Extrude...**.
- Or, on the Features toolbar, choose **Extruded Cut**.

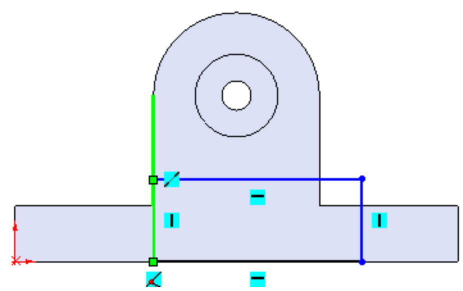
27 Rectangle.
Press the spacebar and double-click `*Front`. Start a sketch on this large face and add a rectangle **Coincident** with the bottom model edge.



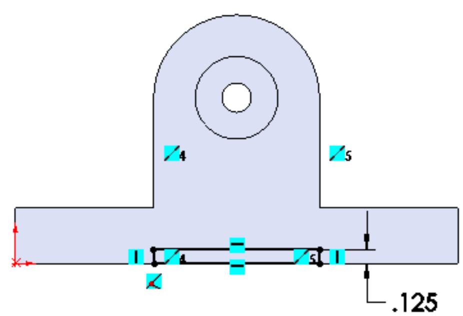
Selecting Multiple Objects

As you learned in Lesson 2, when selecting multiple objects, hold down the **Ctrl** key and then select the objects.

28 Relations.
Select the left vertical sketch line and the vertical model edge. Add a **Collinear** relation between them. Repeat the process on the opposite side.

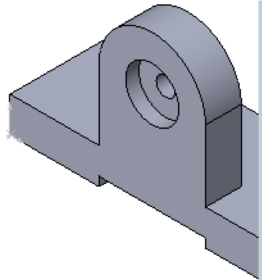


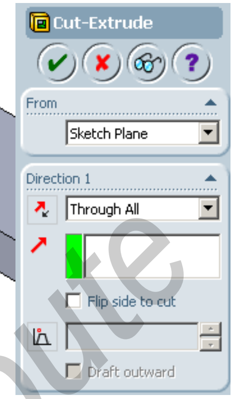
29 Dimension.
Add a dimension to fully define the sketch. Change the view orientation to `Isometric`.



30 Through All Cut.

Click **Insert, Cut, Extrude** or pick the **Extruded Cut** tool on the Features toolbar. Choose **Through All** and click **OK**. This type of end condition always cuts through the entire model no matter how far. No depth setting was needed. Rename the feature `BottomSlot`.



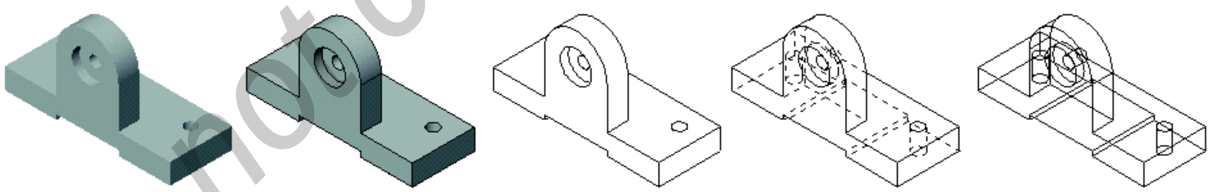


View Options

SolidWorks gives you the option of representing your solid models in one of several different ways. They are listed below, with their icons:

- **Shaded**
- **Shaded with Edges**
- **Hidden Lines Removed**
- **Hidden Lines Visible**
- **Wireframe**

Examples of each are shown in the illustration below. You will learn more about view display and manipulation in *Lesson 4: Modeling a Casting or Forging*.



Shaded | Shaded with Edges | Hidden Lines Removed | Hidden Lines Visible | Wireframe

Filleting

Filleting refers to both fillets and rounds. The distinction is made by the geometric conditions, not the command itself. Fillets are created on selected edges. Those edges can be selected in several ways. Options exist for fixed or variable radius fillets and tangent edge propagation.

Both fillets (adding volume) and rounds (removing volume) are created with this command. The orientation of the edge or face determines which is used.

Filleting Rules

Some general filleting rules are:

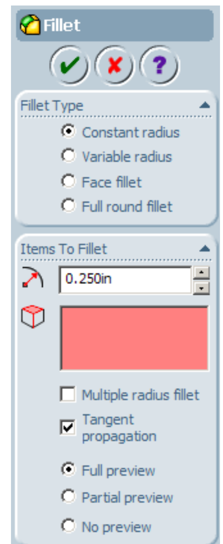
1. Leave cosmetic fillets until the end.
2. Create multiple fillets that will have the same radius in the same command.
3. When you need fillets of different radii, generally you should make the larger fillets first.
4. Fillet order is important. Fillets create faces and edges that can be used to generate more fillets.

Where to Find It

- From the **Insert** menu, select **Features, Fillet/Round...**.
- Click the tool on the Features toolbar.

31 Insert Fillet.

Select the **Fillet** option in one of the ways mentioned above. The **Fillet** options appear in the PropertyManager. Set the radius value.



- (Radius) = **0.25"**

Preview

You have a choice between **Full preview**, **Partial preview** and **No preview** of the fillet. **Full preview**, as shown below, generates a mesh preview on each selected edge. **Partial preview** only generates the preview on the first edge you select. As you gain experience with filleting, you will probably want to use **Partial** or **No preview** because they are faster.

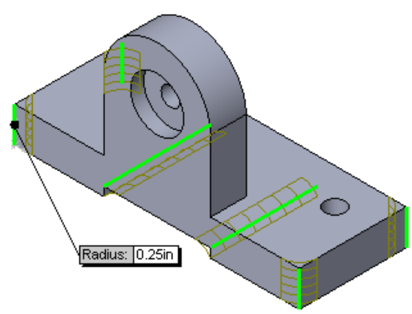
Tip

The display can be changed to **Hidden Lines Visible** to make it easier to select the edges. The edges can be selected "through" the shaded model as displayed below.

32 Edge selection.

The edges will highlight red as the cursor moves over them and then appear green as they are selected. Edges are automatically filtered by the **Fillet** command.

A callout Radius: 0.25in appears on the first edge you select. Select six edges total and click **OK**.

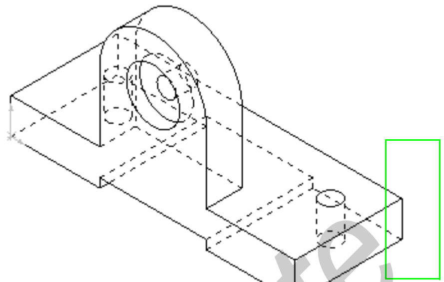


A Note About Color

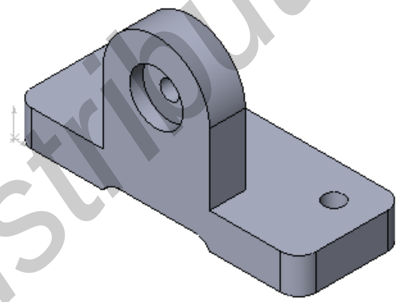
You can customize the colors of the SolidWorks user interface. This is done through **Tools, Options, System Options, Colors**. You can select predefined color schemes, or create your own. In some cases, we have altered colors from their default settings to improve clarity and reproduction quality. As a result, the colors on your system may not match the colors used in this book.

Tip

You can also select edges using a window. Using the left mouse button, drag a window surrounding one or more edges. Edges that are entirely inside the window are selected.

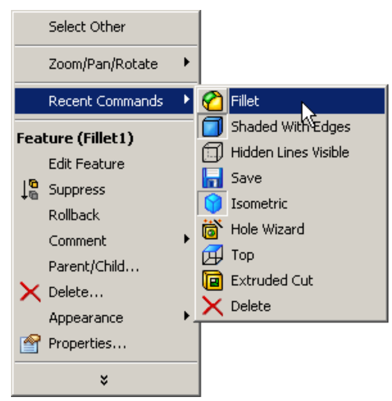


33 Completed fillets.
All six fillets are controlled by the same dimension value. The creation of these fillets has generated new edges suitable for the next series of fillets.



Recent Commands Menu

SolidWorks provides a "just used" buffer that list the last few commands for easy reuse.



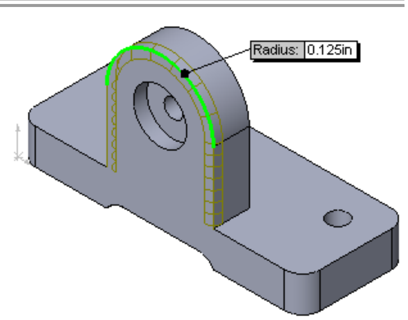
34 Recent Command.
Right-click in the graphics window and select **Recent Commands** and the **Fillet** command from the dropdown list to use it again.

Fillet Propagation

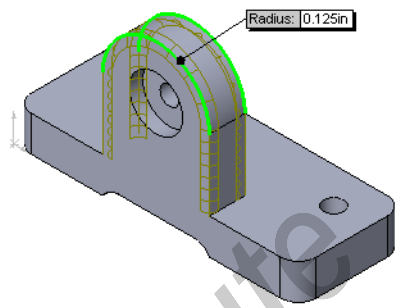
A selected edge that connects to others in a smooth fashion (through tangent curves) can propagate a single selection into many.

35 Preview and propogate.
Add another fillet, radius **0.125"**, using **Full preview**.

Select the edge indicated to see the selected edges and preview.



36 Additional selections.
Select the inner arc edge to see another preview with propagation.

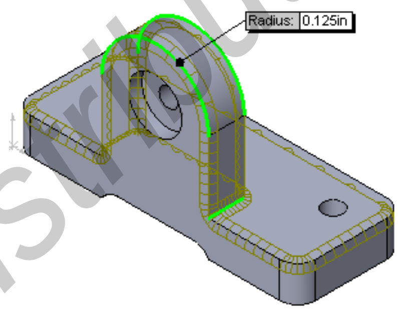


Note

A callout only appears on the first edge you select.

37 Last selection.
Select one final edge to complete the fillet. More propagation occurs due to the connections between the edges.

Click **OK**.



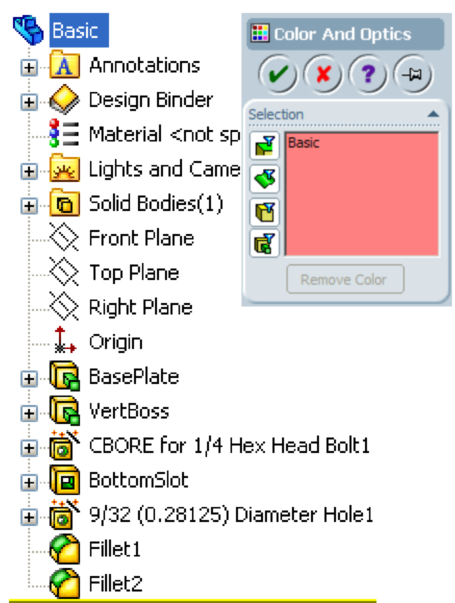
Introducing: Edit Color

Use **Edit Color** to change the color and optical properties of graphics. Color **Swatches** can also be created for user defined colors.

Where to Find It

- Click **Edit Color** on the Standard toolbar.
- Or, right-click a feature, face, surface or body and choose **Appearance, Color**.

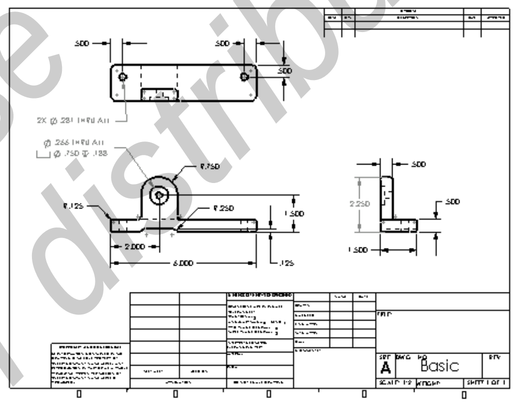
38 Edit Color.
Right-click the FeatureManager top level feature, `Basic`, and click **Appearance, Color**.



39 Select Swatch.
Select the `shiny` swatch and one of the colors. Click **OK**.

40 Save the results.
Click **Save** on the Standard toolbar, or click **File, Save** to save your work.

Detailing Basics



SolidWorks enables you to easily create drawings from parts or assemblies. These drawings are fully associative with the parts and assemblies they reference. If you change the model, the drawing will update.

Various topics related to making drawings are integrated into several lessons throughout this book. The material presented here is just the beginning. Specifically:

- Creating a new drawing file and sheet
- Creating Model and Projected drawing views
- Inserting model dimensions
- Adding driving (model) dimensions
- Adding annotations

A comprehensive treatment of detailing is offered in the course *SolidWorks Essentials: Drawings*.

Settings

Settings are accessed through **Tools, Options**. The settings used in this lesson are:

System Options	Document Properties (Set using drawing template)
Drawings, Display Style: • **Display style for new views = Hidden lines visible** • **Tangent edges in new views = Removed**	Detailing: • **Dimensioning standard = ANSI** • **Automatic update of BOM = Selected** • **Auto insert on view creation:** ❑ **Center marks = Selected** ❑ **Centerlines = Cleared** ❑ **Balloons = Cleared** ❑ **Dimensions marked for drawing = Cleared**
Colors: • **Drawings, Hidden Model Edges = Black**	Detailing, Annotations Font, Dimension: • **Font = Century Gothic** • **Height = 12pt**
	Detailing, Dimensions: • **Precision, Primary Units = .123**
	Units = Inches

Toolbars

There are toolbars that are specific to the process of detailing and making drawings. They are:

- **Drawing**

- **Annotation**

New Drawing

Drawing files (`*.SLDDRW`) are SolidWorks files that contain drawing sheets. Each sheet is the equivalent of a single sheet of paper.

Introducing: Make Drawing from Part

Make Drawing from Part takes the current part and steps through the creation of a drawing file, sheet format and initial drawing views using that part.

Where to Find It

- Click **Make Drawing from Part/Assembly** on the Standard toolbar.
- Or, click **File, Make Drawing from Part.**

1 Create Drawing.
Click the **Create Drawing from Part/Assembly** icon and choose `A-Scale1to2` from the **Training Templates** tab.

The sheet format creates an `A-Landscape` drawing. This is an A-size drawing ($8^1/_2$” x 11”) arranged with its long edge horizontal. The sheet format includes a border, title block, and other graphics.

Tip Double-clicking the template will automatically open it, eliminating the need to click **OK**.

Drawing Views

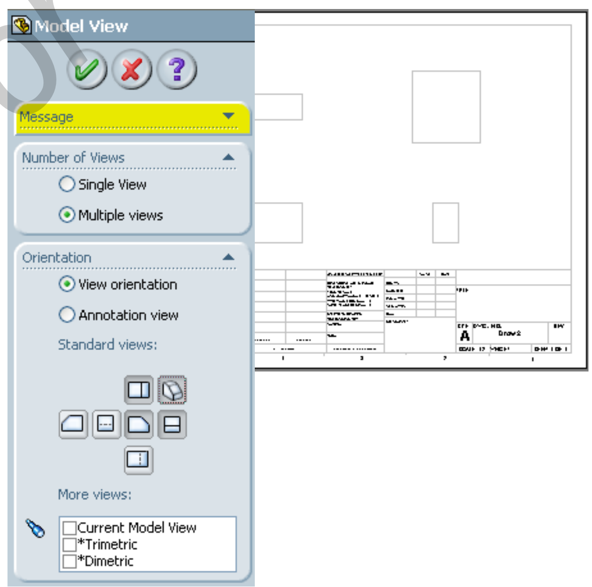
The initial task of detailing is the creation of views. Using the Make drawing from part tool leads you through the drawing sheet to the creation of **View Orientation** views. The View Orientation option creates drawing views that match the orientations in the part.

These options are discussed in detail in the *SolidWorks Essentials: Drawings* manual.

2 Drawing views.
Click **Multiple views**, **View orientation** and select the four standard views (`Front`, `Top`, `Right` and `Isometric`) as shown.

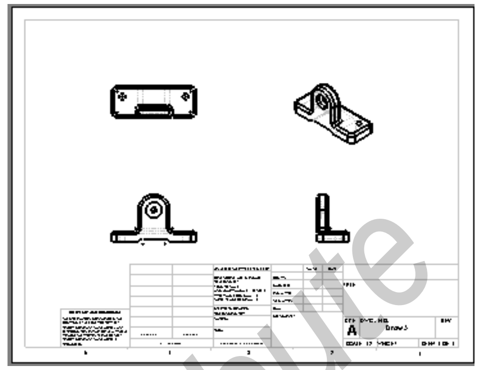
On the **Display Style** tab, click the **Hidden Lines Visible** button.

Click **OK** to create the drawing views.



Note The drawing sheet can be any color. The color is used here to distinguish the part from the drawing.

3 Drawing views.
The drawing views are created on the drawing.

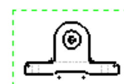


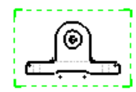
Tip Use **Ctrl-drag** to break the default angled alignment and drop the isometric view anywhere on the drawing.

Set the **Display Style** for this view to **Shaded With Edges**.

Tip The part document is still open. You can press **Ctrl+Tab** to switch between the drawing and part document windows.

4 Tangent edges.
Select inside the a drawing view, between the model and the temporary dotted border, to display the view border. Double-clicking locks the view focus on that view.





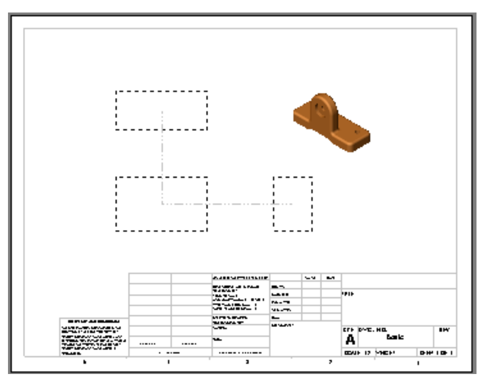
Right-click in the `Front` view and choose **Tangent Edge, Tangent Edges Removed**. Repeat for the `Top` and `Right` views.

5 Display style.
Select the `Isometric` view and change the **Display Style** to **Shaded**.

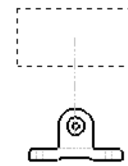
Moving Views

Drawing views can be repositioned by dragging them around the drawing. In the standard 3 view arrangement, the `Front` view is the *source* view. This means that moving the front view moves all three views. The `Top` and `Right` views are *aligned* to the `Front`. They can only move along their axis of alignment.

6 Move Aligned Views.
Select and move the `Front` view. It can be moved in any direction and the other views remain aligned.

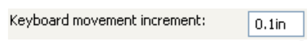


Moving one of the projected views is limited by the alignment.



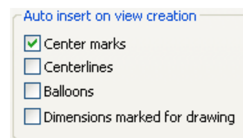
Tip Use **Alt-drag** to select anywhere in the view. Use **Shift-drag** to maintain the spacing between the views while dragging.

Note Once the drawing view has been selected, it can be dragged with the mouse or moved with the arrow keys. The distance moved for each press of an arrow key is set under **Tools**, **Options**, **System Options**, **Drawings**, **Keyboard movement increment**.

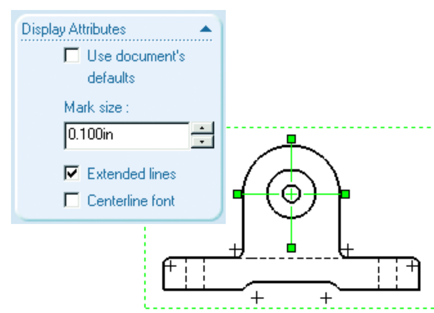


Center Marks

Center marks were inserted into the drawing views automatically. You can turn this option on or off. Set your preference using the **Tools, Options, Document Properties, Detailing** menu.



7 **Center Mark Properties.**
Click on the center mark on the circle in the front view. Check the **Extended lines** option.



Model Dimensions

Model dimensions are simply dimensions and parameters that were used to create the part and that have been inserted into the drawing. These dimensions are considered to be *driving* dimensions. Driving dimensions can be used to make *changes* to the model. You can insert model dimensions into the drawing in four ways. You can automatically insert all the dimensions associated with:

- **A selected view**
- **Selected feature(s)**
- **Selected components in an assembly**
- **All views**

Inserting All Model Dimensions

The dimensions created in the part will be used in the detail drawing. In this case all the dimensions in all views will be inserted. When the system inserts model dimensions into all views, it starts with any detail and section views first. Then it adds any remaining dimensions to remaining views based on which views are most appropriate for the features being dimensioned.

Introducing: Insert Model Items

Insert Model Items allows you to take the dimensions that were created while modeling and insert them into the drawing. Dimensions imported from the model can be used to change the model. These dimensions are called *driving* dimensions.

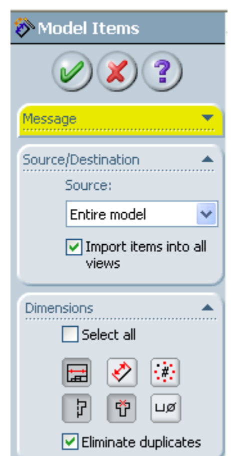
Where to Find It

- From the menu select **Insert, Model Items...**.
- Or, on the Annotations toolbar, click .

---

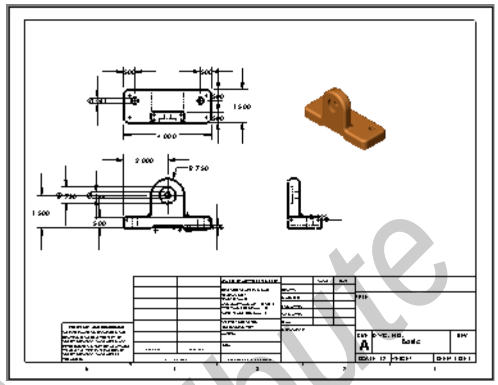
8 Insert Model Items.
Click **Insert, Model Items** and **Import from** the **Entire model**.

Click the options for **Marked for drawing** and **Hole Wizard Locations** and click **Import items into all views**.



Tip

The **Marked for drawing** option selects those dimensions which were marked in the part. The marking option appears in the **Modify** tool where dimension values are set. By default, all dimensions are marked for import into the drawing. Unmarked dimensions appear with blue text.



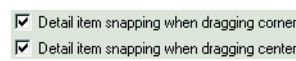
9 Resulting Dimensions.
The dimensions are added into the drawing, but generally not at their final locations. Placing dimensions carefully in the model when you sketch will save time when they are imported into the drawing.

Once the dimensions are inserted, they are associated to that view and will move with it unless you deliberately move them to another view or delete them.

Manipulating Dimensions

Once dimensions have been added to a view, there are several options as to how they can be manipulated:

- **Drag them into position.**
  Drag dimensions by their text to new locations. Use the inference lines to align and position them.

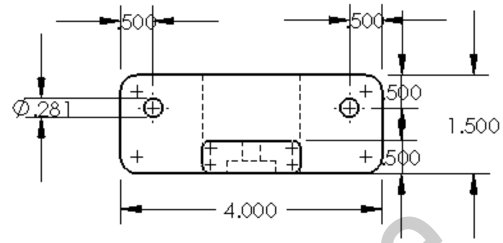
  To facilitate positioning dimensions, the **Drawings** settings on the **Tools, Options, System Options** dialog box has two **Detail item snapping** options. The inferences are displayed when you drag a dimension or note by its center or corner.

- **Hide them.**
  Some dimensions created in the model have limited use in the drawing so you might want to hide them. Right-click the dimension text and select **Hide** from the shortcut menu. The dimension will be removed from the drawing sheet, but not from the model's database.

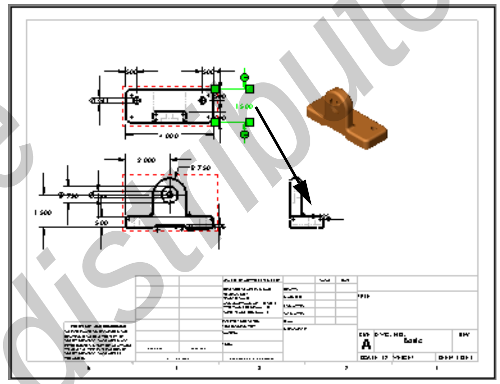
- **Move or Copy them to other views.**
  Many times a feature can be dimensioned in more than one view. The dimension may not automatically appear in the view where you want it. You can move dimensions between views as long as the destination view can display that dimension.

  To move a dimension hold down **Shift** and drag the dimension to another view. To copy the dimension, hold down **Ctrl** and drag it into another view and drop it.

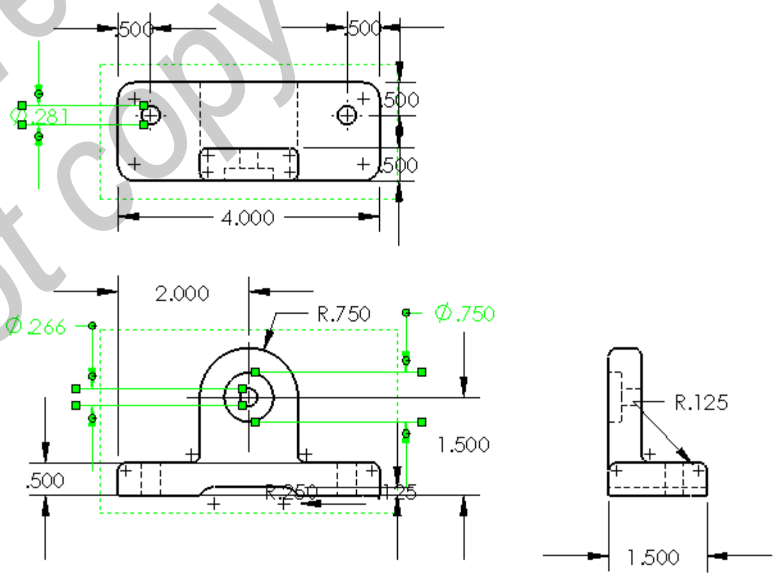
10 Repositioning dimensions.
The top view contains several dimensions. Some of them will be repositioned into the right view.



11 Moving a dimension.
In this case, move the **1.500** dimension from the top view to the right side view using the Shift-drag technique.



12 Delete dimensions.
Delete the diameter dimensions shown. They will be replaced by **Hole Callouts**, a type of annotation that is a driven dimension.

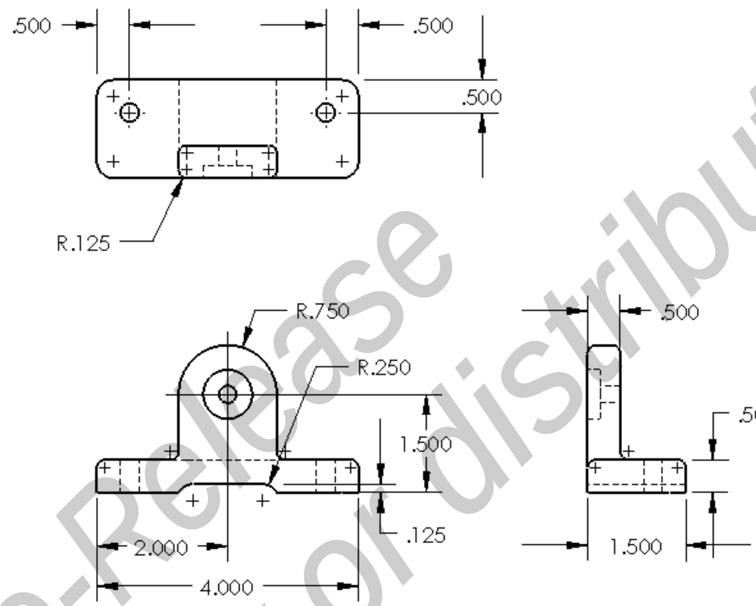


Note

Deleting a dimension in the drawing *does not* delete it from the model. Deleted dimensions can be re-inserted from the model.

13 Dimensions after moving and deleting.

The illustration below shows the result of moving several other dimensions into the right side view. It also shows the results of rearranging the dimensions in the top view.



Driven Dimensions

Not all of the dimensions you need on a drawing may be present in the model. Sometimes, because of the way the model was built, there is no dimension to import into the drawing. When this happens, the dimension has to be inserted manually using the same dimension tool you use when sketching. This type of dimension is called a *driven* dimension because its value is driven by the model. Unlike driving dimensions, you cannot change its value and thereby change the model.

Dimension Display

By default, driven dimensions are displayed differently than driving dimensions:

The value is enclosed in parentheses. This is accepted practice for reference dimensions.

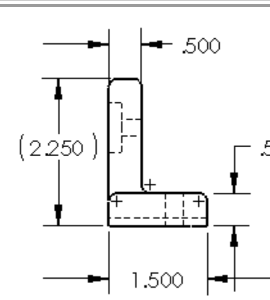
They are displayed in a different color, in this case, gray.

14 Dimensioning.

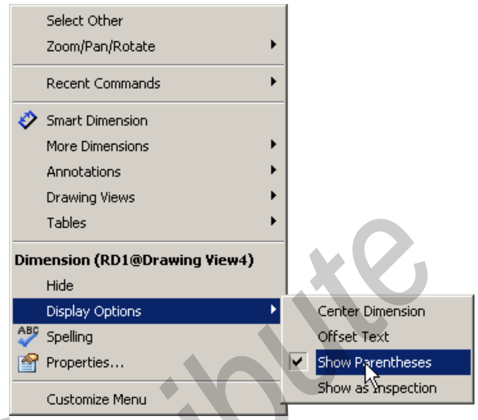
Click **Vertical Dimension** .

Dimension the overall height of the vertical boss.

Click the tool again to turn it off.



15 Display options.
The appearance of a dimension can be changed in many ways. Right-click the dimension and clear the **Display Options, Show Parentheses** option.



Associativity Between the Model and the Drawing

In the SolidWorks software, everything is associative. If you make a change to an individual part, that change will propagate to any and all drawings and assemblies that reference it.

Procedure

To change the size of the `BasePlate` feature follow this procedure:

16 Switch windows.
Press **Ctrl+Tab** to switch back to the part document window.

Changing Parameters

SolidWorks mechanical design automation software makes it very easy to make changes to the dimensions of your part. This ease of editing is one of the principal benefits of parametric modeling. It is also why it is so important to properly capture your design intent. If you don't properly capture the design intent, changes to dimensions may cause quite unexpected results in your part.

Rebuilding the Model

After you make changes to the dimensions, you must rebuild the model to cause those changes to take affect.

Rebuild Symbol

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

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

Introducing: Rebuild

Rebuild regenerates the model with any changes you have made.

Where to Find It

- Click **Rebuild** on the Standard toolbar.
- Or, on the **Edit** menu, click **Rebuild**.
- Use the keyboard shortcut **Ctrl+B**.

Refreshing the Screen

If you simply want to refresh the screen display, removing any graphic artifacts that might remain from previous operations, you should use **Redraw**, not **Rebuild**.

Introducing: Redraw

Refreshes the screen, but does not rebuild the part.

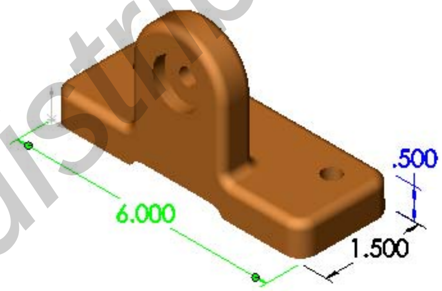
Where to Find It

- From the **View** menu, click **Redraw**.
- Use the keyboard shortcut **Ctrl+R**.

Rebuild vs. Redraw

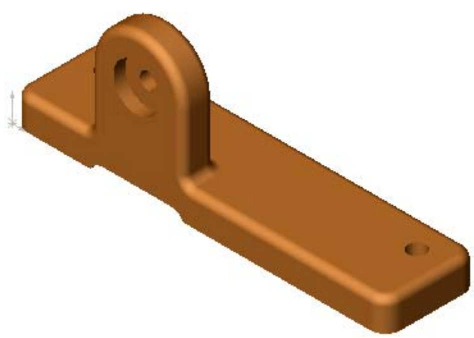
Redraw will *not* cause changes to dimensions to take affect. Therefore, it is very fast. **Rebuild** regenerates the model. Depending on the complexity of the model, this can take more time.

17 Double-click on the feature.
You can double-click on the `BasePlate` feature either in the FeatureManager design tree or the graphics window. When you do this, the parameters associated with the feature will appear.

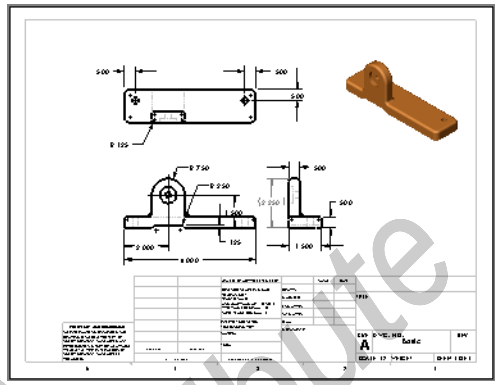


Double-click on the **4** inch dimension indicated. The **Modify** dialog box will appear. Enter a new value either by typing it directly or by using the spin box arrows. Enter **6** inches.

18 Rebuild the part to see the results.
You can **Rebuild** the part either by clicking on the **Rebuild** tool on the **Modify** box or on the Standard toolbar. If you use the one on the **Modify** dialog box, the dialog box will stay open so you can make another change. This makes exploring “what if” scenarios easy.



19 Update the drawing.
Switch back to the drawing sheet. The drawing will update automatically to reflect the changes in the model.



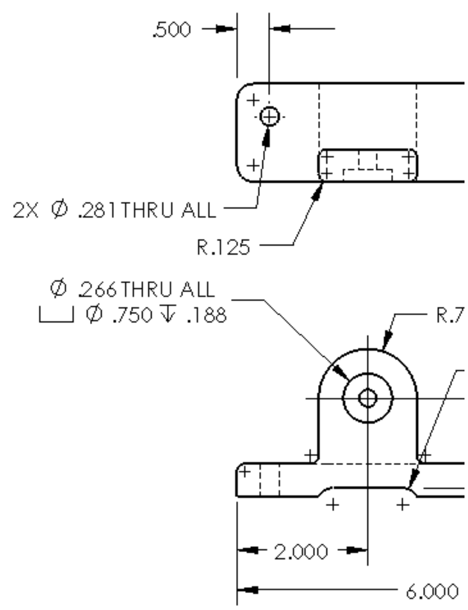
Introducing: Hole Callouts

The **Hole Callout** tool is used to add driven diameter dimensions to holes created by the **Hole Wizard** or circular cut features. It is one of many annotations available in SolidWorks.

These annotations can be added automatically using **Insert Model Items**.

- Click **Insert, Annotations, Hole Callout**.
- Or on the Annotations toolbar, click **Hole Callout**.
- Or right-click and select **Annotations, Hole Callout**.

20 Add Hole Callouts.
Click the center hole in the front view and place the annotation on the drawing. Select the left of the two holes in the top view and place.



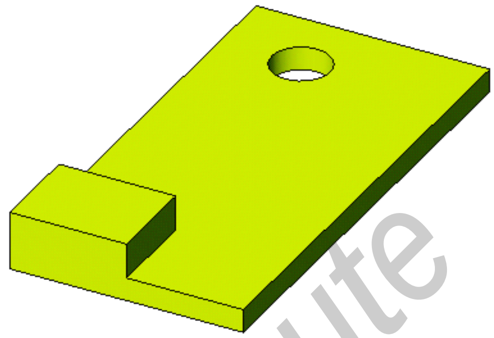
Note

The **“2X”**prefix is added automatically because there are two drilled holes.

21 Save and close the part and drawing.

Exercise 4: Plate

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part. This lab reinforces the following skills:



- Sketching
- Base Extrusion
- Boss Extrusion
- Hole Wizard

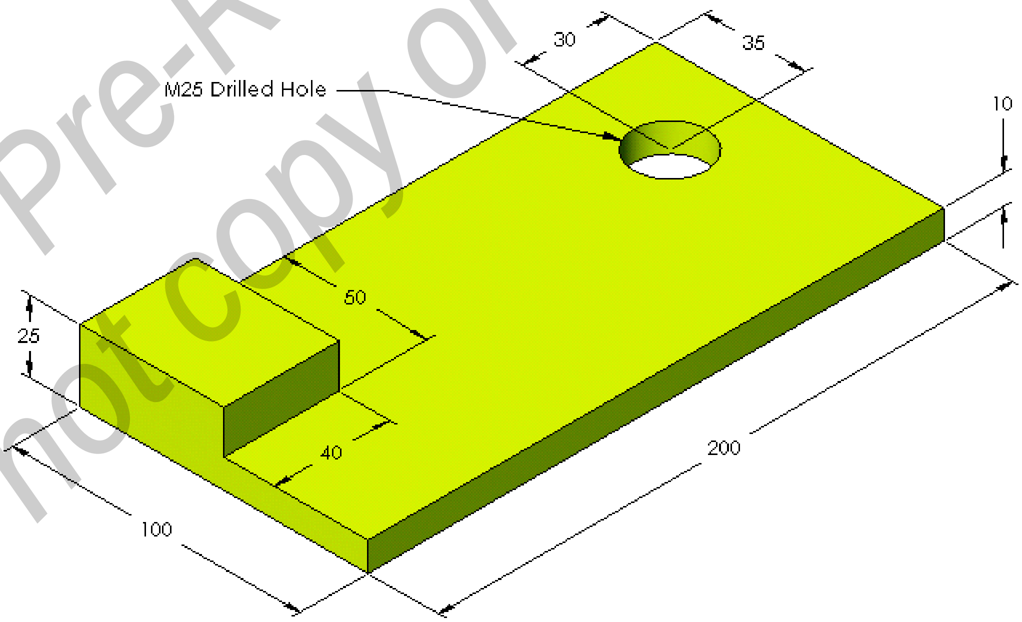
Design Intent

Use the design intent to create the part.

1. The part is not symmetrical.
2. The hole is an ANSI Metric Drill Size hole.

Dimensions

Use the following graphics with the design intent to create the part.

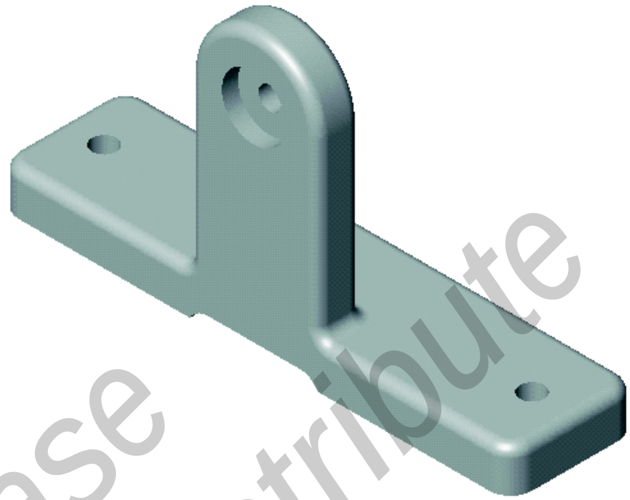


Exercise 5: Basic-Changes

Make changes to the part created in the previous lesson.

This exercises uses the following skills:

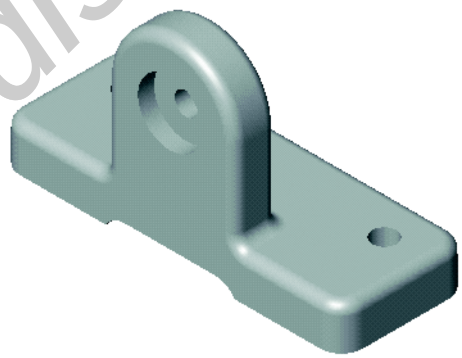
- Changing Dimension Values.



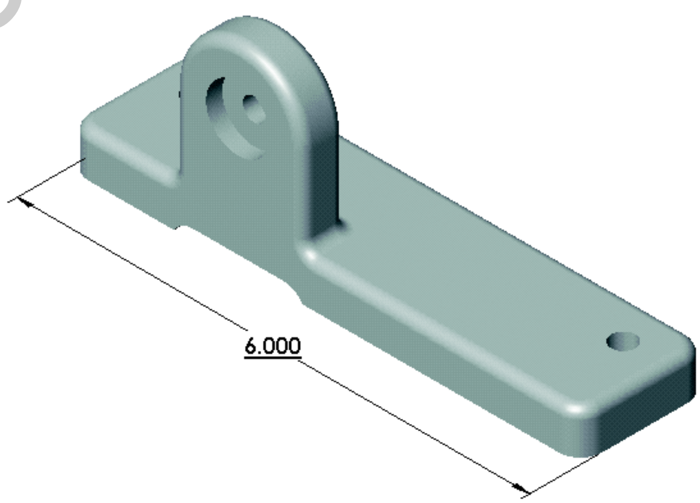
Procedure

Open an existing part in the `Exercises` folder.

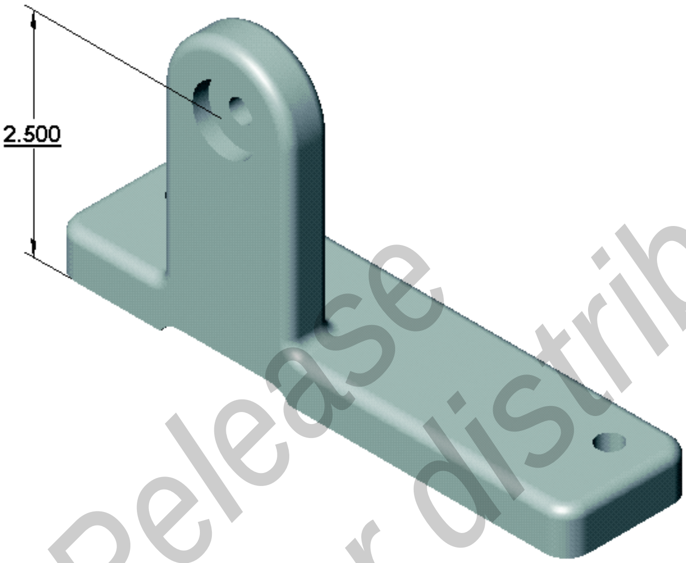
1 **Open the part `Basic-Changes`.**
Several changes will be performed on the model to resize it and check the design intent.



2 **Overall dimension.**
Double-click the first feature (`Base Plate`) in the FeatureManager or on the screen to access the dimensions. Change the length dimension to **6in** (shown bold and underlined below) and rebuild the model.

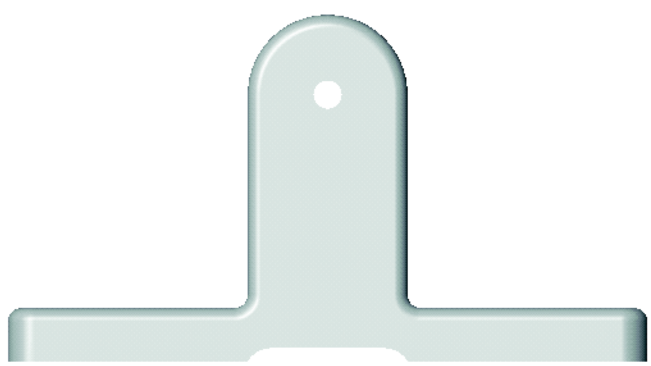


3 Boss.
Double-click the `Vert boss` feature and change the diameter and height dimensions as shown. Rebuild the part.



4 Hole locations.
Double-click the `9/32 Holes` feature and change the position dimensions to **0.75in** each. Rebuild the model.

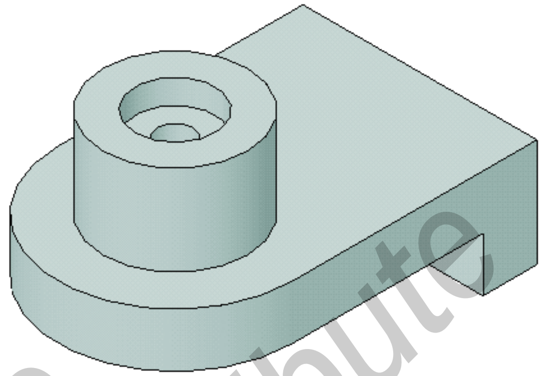
5 Center the `Vert Boss`.
Determine the proper value and change the dimension that centers the `Vert Boss` on the base.



6 Save and close the part.

Exercise 6: Bracket

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.



This lab reinforces the following skills:

- Sketching.
- Bosses.
- Holes.

Design Intent

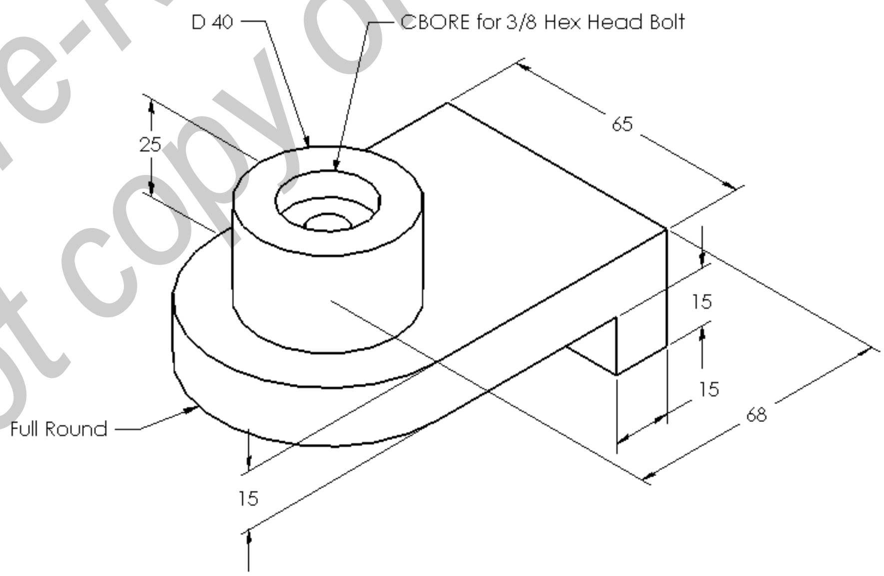
The design intent for this part is as follows:

1. The boss is centered on the rounded end of the base.
2. The hole is a through hole and is concentric to the boss.

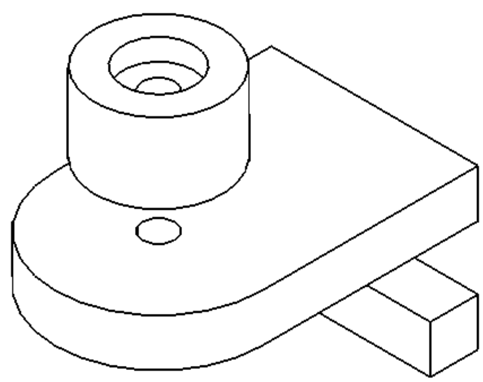
Use the `Part_MM` template.

Dimensioned View

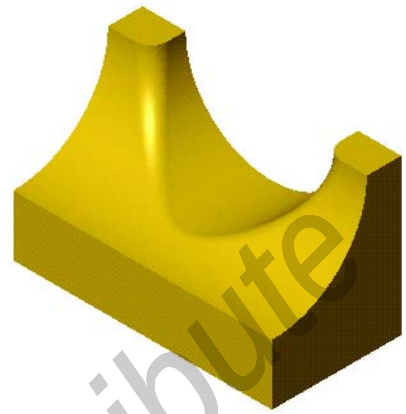
Use the following graphics and the design intent to create the part.



As an aid to constructing this part, visualize how it could be broken down into individual features:



Exercise 7: Working with Fractions



Create this part using the information and fractional dimensions provided. Sketch and extrude profiles to create the part. This lab reinforces the following skills:

- Entering and displaying dimensions as fractions.
- Bosses.
- Cuts.
- Fillets.
- **Blind** and **Through All** end conditions.

Fractions

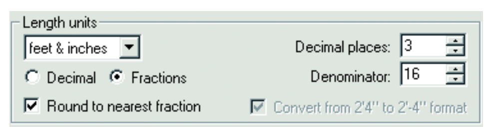
There are two things to consider when working with dimensions that are given in fractions:

1. Setting the document units to fractional inches.
2. Entering dimension values as fractions.

Document Units

On the **Tools, Options** dialog, click the **Document Properties** tab and select **Units**. The two types of length units that support **Fractions** are:

- **Inches**
- **Feet & Inches**



When you choose **Fractions**, you should specify the default **Denominator**. Dimensions that are evenly divisible by this denominator are displayed as fractions. How dimensions are displayed that are *not* evenly divisible depends on whether you select the **Round to nearest fraction** option.

For example, if the **Denominator** is set to 16 and you enter a value of 3⁄64 the value will display as 1⁄16 if **Round to nearest fraction** is selected. It will display as 0.047 if it is not selected.

Entering Dimensions

You can enter dimensions as fractions regardless of whether the document units are set to fractions. To enter a value such as 1 7⁄8”, type 1, press the Spacebar, then type 7/8, and press Enter.

Design Intent

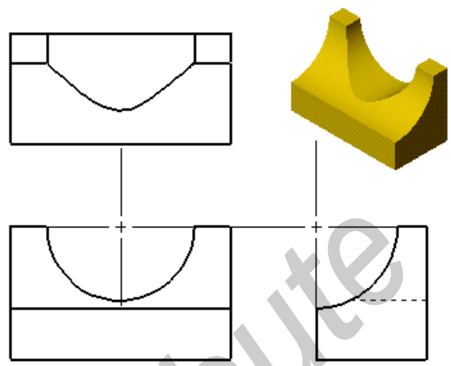
The design intent for this part is as follows:

1. The side to side cut is centered on the corner.
2. The front to back cut is centered at the midpoint of the edge.

Use the `Part_IN` template.

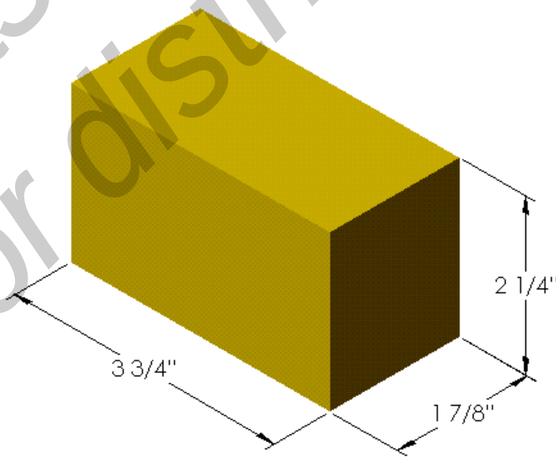
Views

Use the following graphics to help visualize the part.

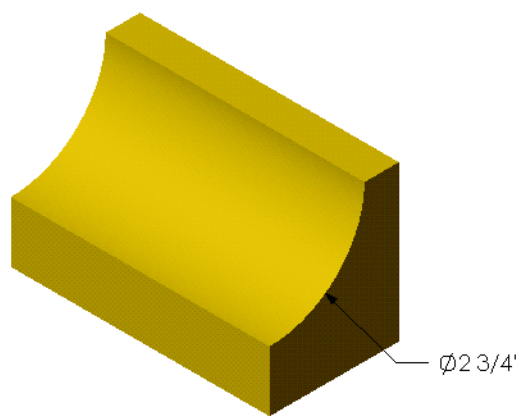


1 **New part.**
Open a new part using the `Part_IN` template.

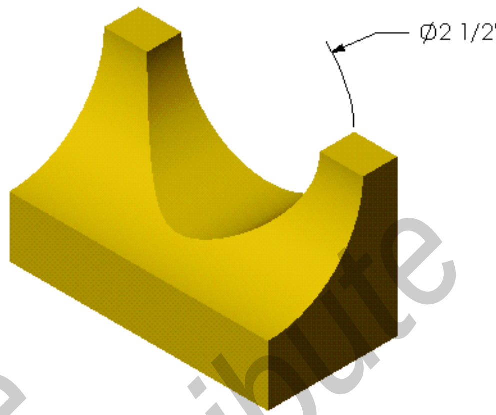
2 **First feature.**
Create the first feature using a sketch and an extruded boss. You can use either the `Front`, `Top` or `Right` reference plane for sketching.



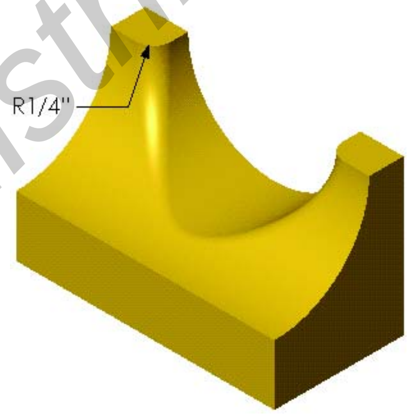
3 **Cut Feature.**
Using a sketch containing a circle, create an extruded cut feature. The circle is centered on the corner (vertex) of the first feature.



4 Second cut feature.
Create a second extruded cut using a circle. This circle should be centered at the midpoint (halfway along the edge).



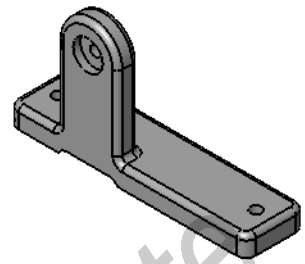
5 Fillet/Round.
Using the edge created by the cuts, create a fillet/round feature.



6 Save and close the part.

Exercise 8: Part Drawings

Create this part drawing using the information provided.



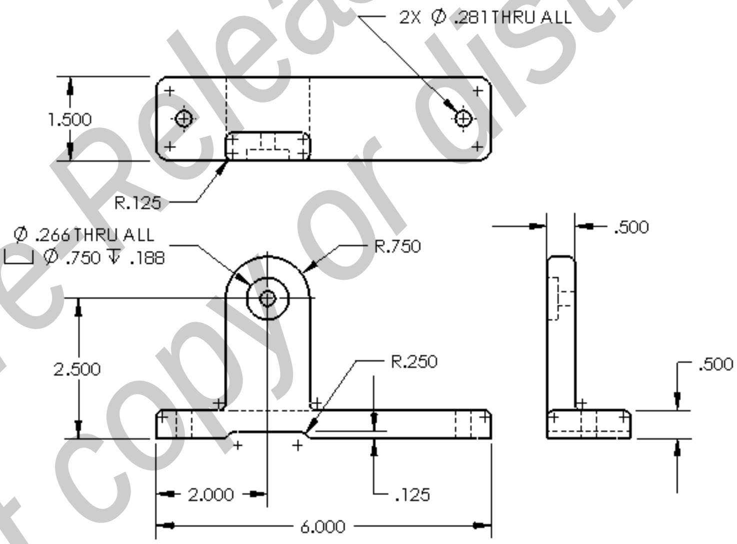
This lab reinforces the following skills:

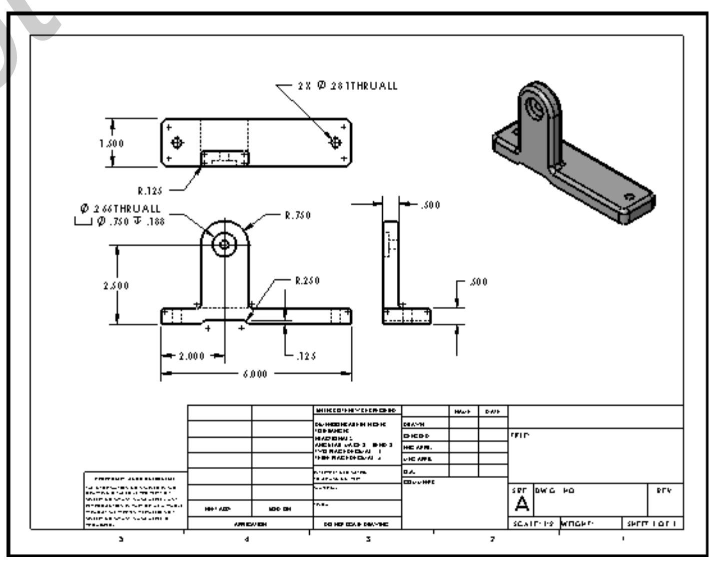
- Drawing Sheets.
- Drawing Views.
- Center Marks.
- Dimensions.
- Hole Callouts.

Use the `A-Scale1to2` template and the built part `Basic-Changes-Done`.

Dimensioned View

Use the following graphics to create the drawing.





Exercise 9: Guide

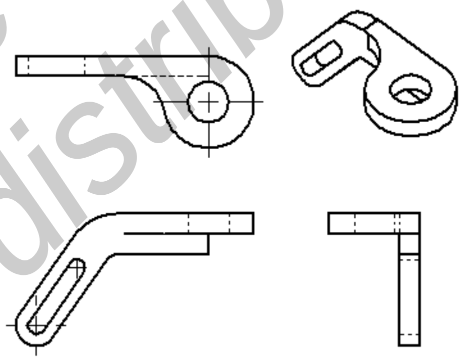
This lab reinforces the following skills:

- Sketch lines, arcs, circles and fillets.
- Relations.
- Extrusions.
- Fillets and rounds.

Design Intent

Some aspects of the design intent for this part are:

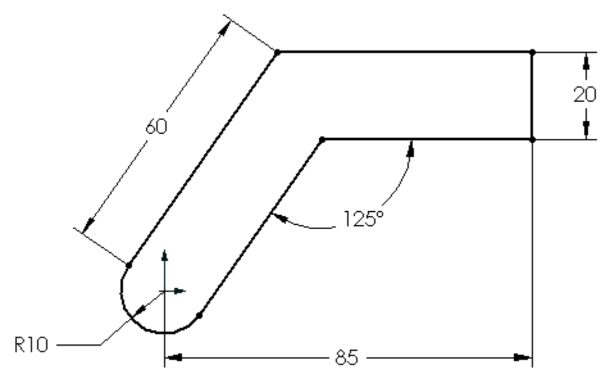
1. Part is not symmetrical.
2. Large circle is tangent to outer edge.
3. Large circle is coincident with underside brace edge.
4. Plate thicknesses are equal.



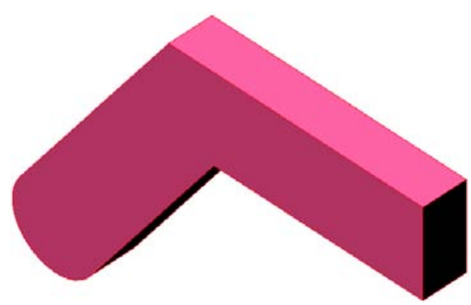
Procedure

Open a new part using the `Part_MM` template.

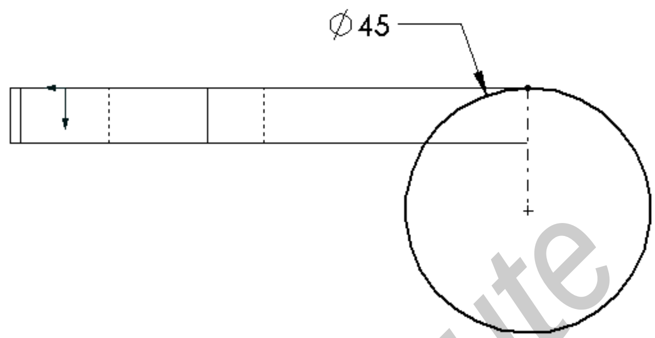
1 Sketch the profile.
Using the `Front` plane, create the profile.



2 Extrusion.
Extrude the sketch **10mm**.

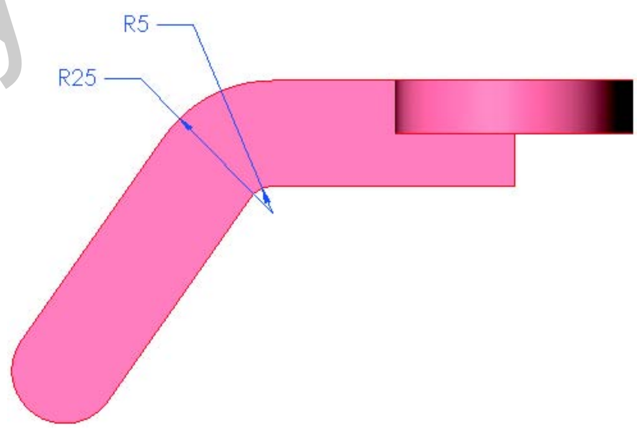


3 Upper sketch.
Start a sketch on the top face of the model. The circle is tangent to one edge and coincident to another edge.

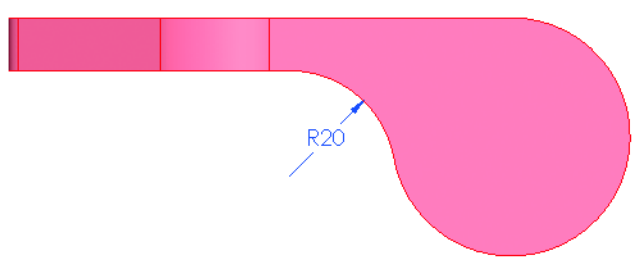


4 Extrude equal thickness.
Extrude the circle the same thickness as the first feature.

5 Add two fillets.
Add two fillets as shown.



6 Last fillet.
Create a third fillet with a **20mm** radius.

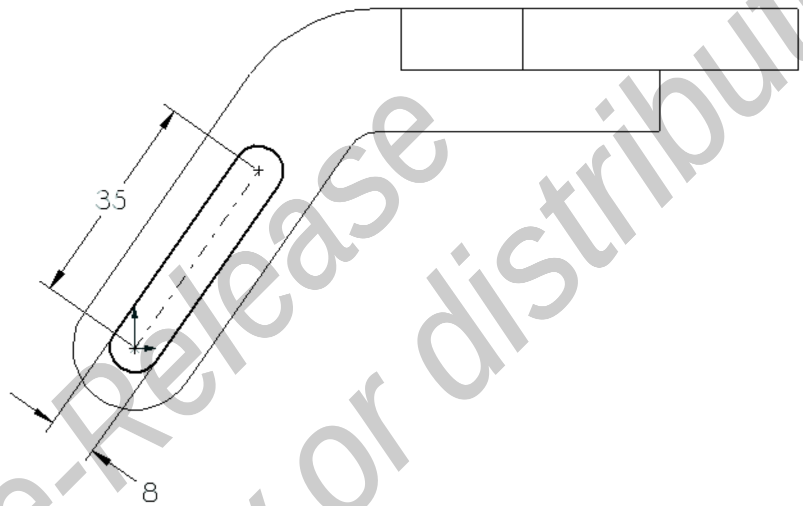


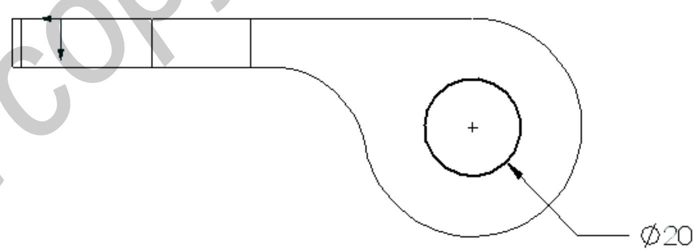
7 Cuts.

Use symmetry with lines and arcs to create a **Through All** cut for the slot shape. Use a circle to create another cut concentric with the model edge.

Note

This sketch requires the use of a **Parallel** relation. Check the **Help, SolidWorks Help Topics** for more information.





8 Save and close the part.

Lesson 4
Modeling a Casting or Forging

Upon successful completion of this lesson, you will be able to:

- Open a SolidWorks part and save your work.
- Sketch on a system defined plane or a planar face of a model.
- Use the view display and modification commands.
- Create fully defined sketches through the use of dimensions and geometric relationships.
- Create base and boss features by extrusion.
- Create cut features by extrusion.
- Copy and paste features.
- Create constant radius fillets.
- Edit the definition and parameters of a feature and regenerate the model.
- Use **Up To Next** and **Mid Plane** end conditions to capture design intent.
- Use symmetry in the sketch.

Case Study: Ratchet

The `Ratchet` contains many of the features and procedures that you will use frequently. It contains bosses, cuts, sketch geometry, fillets and draft.

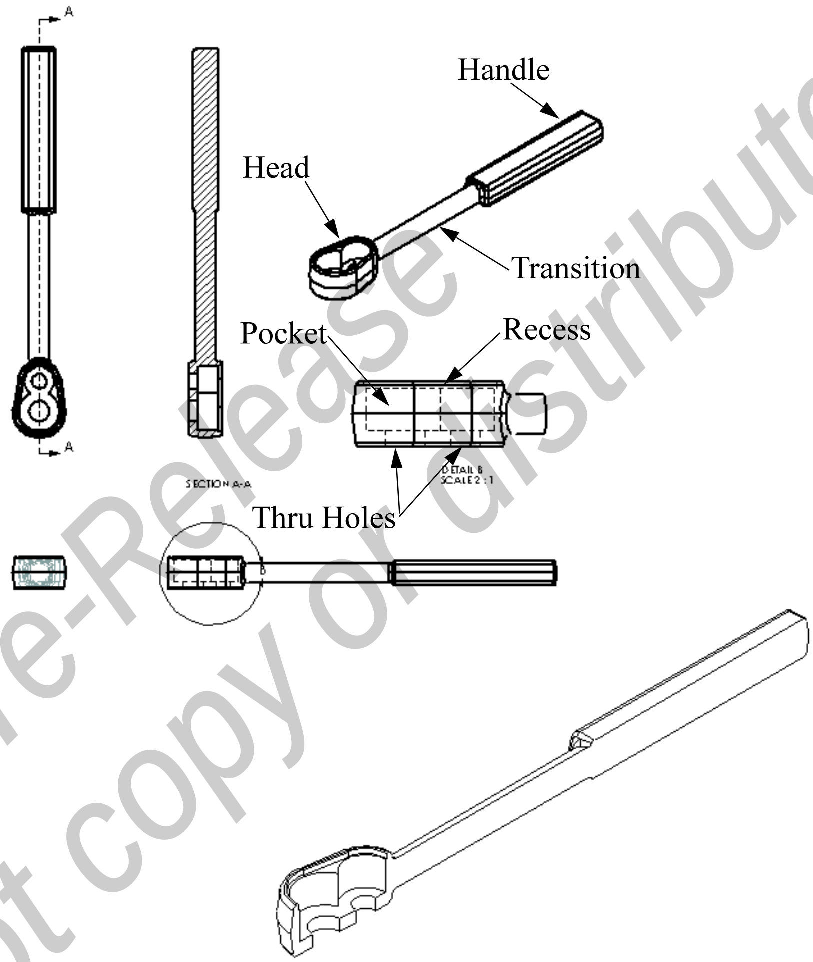
Stages in the Process

Some key stages in the modeling process of this part are shown in the following list. Each of these topics comprises a section in the lesson.

- **Design intent**
  The overall design intent for the part is discussed.
- **Boss feature with draft**
  The first portion of the model to be created is the `Handle`. The `Handle` uses sketched lines and is extruded in two directions with draft forming a solid. It is the initial feature of the part and demonstrates the use of mirroring in the sketch.
- **Up To Next end condition**
  The second portion of the model is the `Transition`. It uses the **Up To Next** end condition to connect to the `Handle`'s faces.
- **Sketching inside the part**
  The third boss created is the `Head`. It is sketched within the solid created by the `Transition`.
- **Cut using existing edges**
  The `Recess` is the first cut type feature created. It uses an offset from the existing edges of the model to create the sketch. It is extruded as a offset cut to a specific depth.
- **Cut with trimmed sketch geometry**
  The `Pocket` is another cut feature, this time using circles that are trimmed to the proper shape.
- **Cut using copy and paste**
  The `Wheel Hole` feature will be copied and pasted.
- **Filleting**
  Fillets and rounds are added to the solid using several different techniques.
- **Editing a feature's definition**
  Features that already exist can be changed using **Edit Feature**. Fillets will be edited in this way.

Design Intent

The general design intent of the `Ratchet` is summarized in the illustration and list below. Specific design intent for each portion of the part is discussed separately.



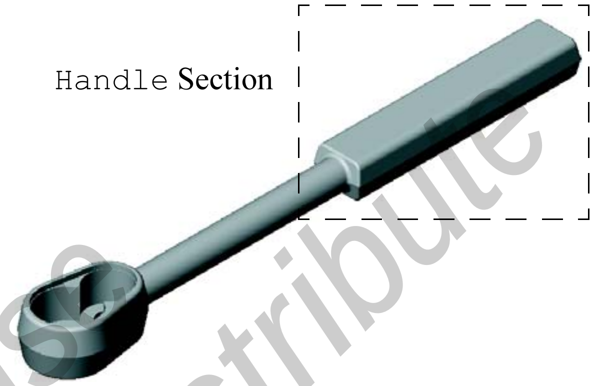
- **Centering:** The `Head`, `Handle` and `Transition` features are centered along an axis.
- **Symmetry:** The part is symmetrical, both with respect to a longitudinal centerline and with respect to the parting plane.

Boss Feature with Draft

The first part of the `Ratchet` we will model is the `Handle`. The first feature in any model is sometimes referred to as the *base* feature. All other features are built onto the first feature.

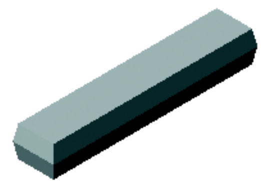
Building the Handle

The `Handle` has a rectangular cross section. It is extruded with draft an equal distance in opposite directions from the sketch plane.



Design Intent of the Handle

The `Handle` is a sketched feature that uses lines and mirroring to form the basic outline or profile. The profile is extruded in opposite directions, equally, with draft. The sketch creates a rectangular cross section that is extruded equally in opposite directions with draft.



- **Draft:** The draft angle is equal on both sides of the parting plane
- **Symmetry:** Feature is symmetrical with respect to parting plane and the centerline axis of the `Handle`

A centerline, a piece of reference geometry, will be used to position and sketch the `Handle` sketch.

The centerline represents distance from the end of the handle to the center of the furthest hole and is also used in mirroring sketch geometry.

Procedure

Begin by following this procedure:

1 **New Part.**
Open a new part using the `Part_MM` template on the `Training Templates` tab. Save the part and name it `Ratchet`.

2 **Display off.**
Toggle the display of relations *off* using **View, Sketch Relations**.

Note

Further lessons will assume that **View, Sketch Relations** is toggled *off*.

3 **Sketch plane.**
Select the reference plane `Top` as the sketch plane. Change the view to a `Top` view.

Introducing: Insert Centerline

Insert Centerline is used to create a reference line in a sketch. The centerline can be vertical, horizontal, or an arbitrary angle depending on how the inferences are used. Because the centerline is considered reference geometry, it does not have to be fully defined in the sketch.

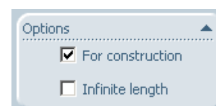
Where to Find It

- Click **Tools, Sketch Entity, Centerline**.
- Or, on the Sketch toolbar click **Centerline**.

Note

Any piece of sketch geometry can be converted into construction geometry or vice-versa. Select the geometry and click the **Construction Geometry** tool on the Sketch toolbar.

The PropertyManager can also be used to change sketch geometry into construction geometry. Select the geometry and click **For construction**.



---

4 **Sketch a centerline.**
Sketch a centerline running vertically from the origin. The length is not important.

Symmetry in the Sketch

Symmetrical geometry in a sketch can be created easily using the **Mirror** option. You can mirror as you sketch – real time mirroring. Or, you can select already sketched geometry and mirror it – after the fact mirroring. Also, **Symmetric** relations can be added to geometry after sketching.

In any case, mirroring creates copies that are related to the originals by the **Symmetric** relation. In the case of lines, the symmetric relation is applied to the endpoints of the lines. In the case of arcs and circles, the symmetric relation is applied to the entity itself. The three methods are listed below.

- **Symmetry while sketching**
- **Symmetry after sketching**
- **Symmetry through relations**

Introducing: Dynamic Mirror

Mirroring requires a line, linear edge or centerline. The line is activated before sketching the geometry to be mirrored.

Where to Find It

- From the **Tools** menu choose: **Sketch Tools, Dynamic Mirror**.
- Or, on the Sketch toolbar click **Dynamic Mirror**.

Symmetry While Sketching

Symmetric geometry can be created in real time as you sketch. The **Dynamic Mirror** method enables mirroring *before* sketching.

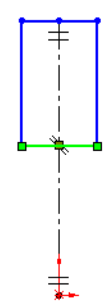
Symmetry after Sketching

Symmetry can be created by sketching one half of the geometry and using mirroring to create the other. The symmetry is applied *after* sketching.

5 **Dynamic mirror.**
Select the centerline and click the **Dynamic Mirror** tool. The **Dynamic Mirror** symbol appears at both ends of the centerline.

6 **Sketch line.**
Sketch a line from the upper end of the centerline moving to the right. A mirror image of the line is created on the opposite side of the centerline.

7 **Complete the sketch.**
Add a line in the vertical direction and then horizontal, stopping at the centerline. Turn off the mirror tool.



Tip

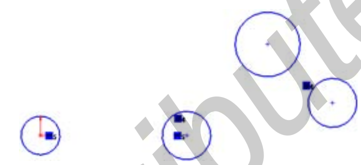
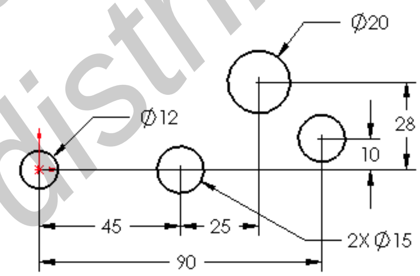
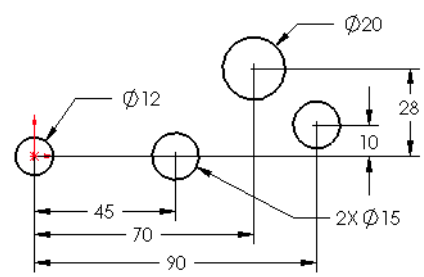
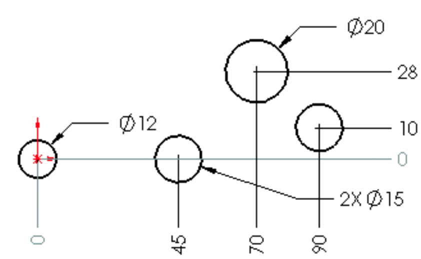
Do not cross the centerline while sketching in the **Automatic Mirror** mode. If you do, duplicate geometry can be created. Stopping at the centerline caused the symmetrical lines to be merged into a single line.

Automatic Dimensioning of Sketches

Autodimension creates dimensions in a sketch. Several dimension styles, such as baseline, chain and ordinate are supported. The starting points for horizontal and vertical sets can be set. This tool does *not* add geometric relations to the sketch.

Introducing: Autodimension

Autodimension has options for dimension type, entities to be dimensioned and starting points.

<table>
<tr><td>Underdefined sketch with geometric relations.</td><td></td></tr>
<tr><td>Chain option selected with start points at origin.
Note: Some dimensions have been moved for clarity.</td><td>
</td></tr>
<tr><td>Baseline option selected with start points at origin.</td><td>
</td></tr>
<tr><td>Ordinate option selected with start points at origin.</td><td>
</td></tr>
</table>

Note

A special option **Centerline** appears when centerline geometry is used in the sketch. Dimensions can be based from the centerline.

Where to Find It

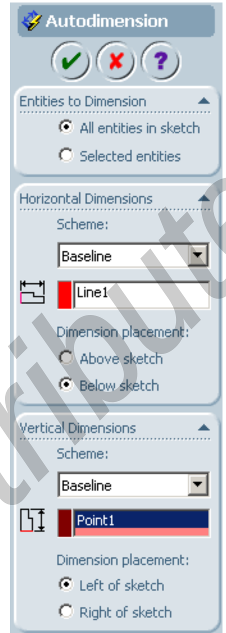
- Click **Tools, Dimensions, Autodimension....**
- Or, on the Dimensions/Relations toolbar, click the **Autodimension** tool.

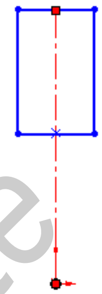
8 Autodimension setup.

Click the **Autodimension** tool.

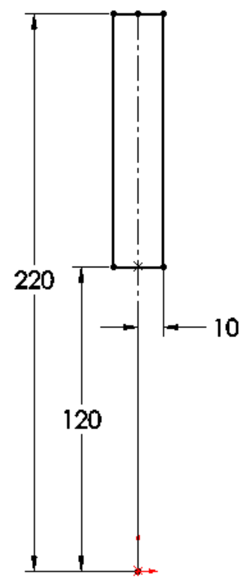
Click in the **Datum** field for the **Horizontal Dimensions** and select the vertical centerline. For the **Vertical Dimensions**, select the endpoint of the centerline.

Set both **Schemes** to **Baseline**.





9 Dimensions.
The dimensions are added, changed and moved for clarity.



Note Sketches dimensioned with Autodimension are fully defined but may not be dimensioned exactly the way you want. You can delete and replace dimensions if required.

First Feature

The first feature, is always a boss, and is the first solid feature created in any part. In this part, the first feature created is a **Mid Plane** extrusion.

10 Base/Boss Extrusion.

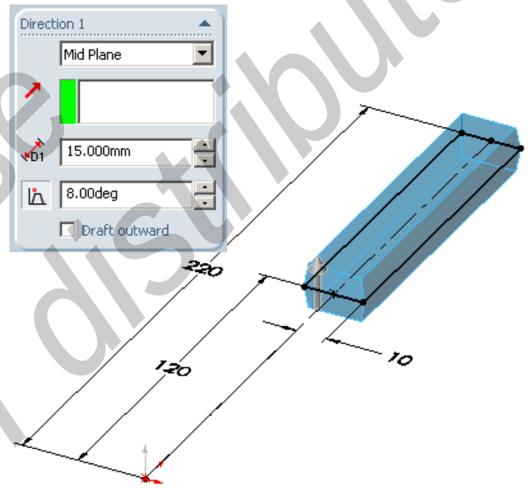
Click the **Extruded Boss/Base** tool on the Features toolbar or click **Boss/Base Extrude** from the **Insert** menu.

---

11 Extrusion.

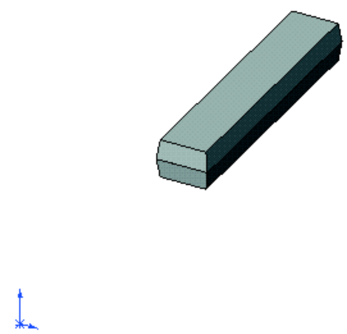
Choose the **Mid Plane** option from the list and enter a depth of **15mm**. Click **Draft** and set the angle to **8°**. The **Draft Outward** check box should be cleared.

Click **OK** to create the feature.



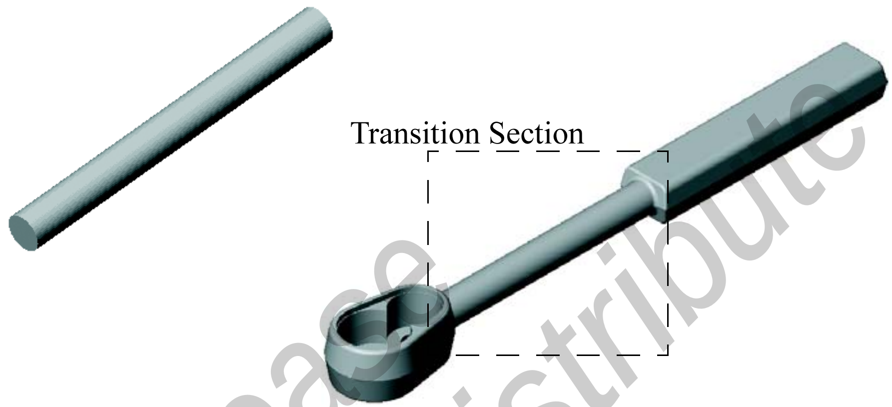
12 Completed feature.

The completed feature is shown at the right. Name the feature `Handle`.



Sketching Inside the Model

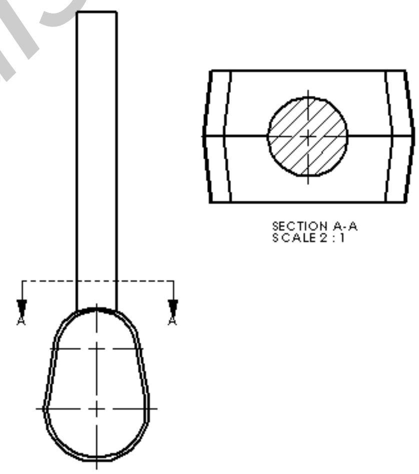
The second feature in the part is the `Transition`, another boss that will connect the `Head` to the `Handle` feature. The sketch for this feature is created on a standard reference plane.



Design Intent of the Transition

The `Transition` feature is a simple circular profile that is extruded up to the existing `Handle` feature.

- **Centering:** The circular profile is centered on the `Head` feature.
- **Length:** The length of the section is determined using existing locations.



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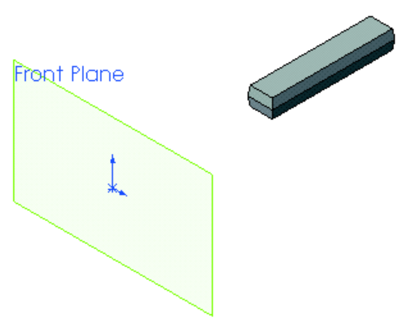
13 Showing the `Front` plane.
Switch to an `isometric` view and select the `Front Plane` from the FeatureManager design tree. It will be highlighted on the screen.

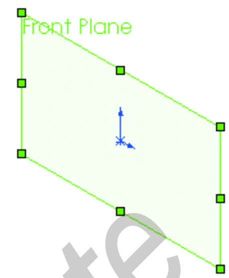


To make sure the plane stays visible, right-click the `Front Plane` in the FeatureManager design tree, and select **Show** from the menu. The plane will appear shaded and transparent.

14 Plane settings and changes.
There are settings to determine how that planes will appear on the screen. For shaded planes, click **Tools, Options, System Options, Display/Selection** and select the **Display shaded planes** check box. Set the color of the plane using **Tools, Options, Document Properties, Plane Display**.

Any plane, system or user generated, can be resized by dragging its handles. Resize this plane so that its borders lie closer to the boundaries of the feature.

The planes can also be automatically sized to the model. Right-click the plane and choose **AutoSize**.



Circular Profile

The sketch for the `Transition` feature has very simple geometry and relations. A circle is sketched and related to a position on the previous feature to define it. This relation will keep the `Transition` centered on the `Handle` feature.

15 Open a new sketch.

With the `Front Plane` still selected click the **Sketch** tool. The plane is now a sketch plane.

Introducing: View Normal To

The **View Normal To** option is used to change the view orientation to a direction normal to a selected planar geometry. The geometry can be a reference plane, sketch, planar face or feature that contains a sketch.

Tip

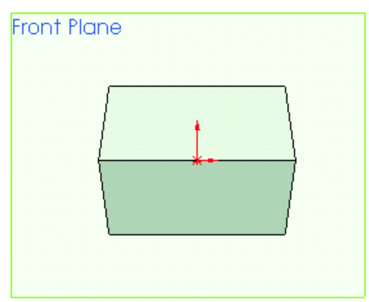
Clicking the **Normal To** icon a second time will flip the orientation around to the opposite side of the plane.

Where to Find It

- Click **Normal To** on the Standard Views toolbar.
- Or, press the **Spacebar** and double-click **Normal To**.

16 Normal To view orientation.

Using the **View Orientation** dialog box, change to the **Normal To** orientation. To do this, select the `Front` plane and double-click the **Normal To** option in the **View Orientation** dialog box. This orients the view so you can see the plane's true size and shape and makes sketching easier.



Tip

You can also select the plane and click the **Normal To** tool on the Standard Views toolbar.

Introducing: Sketched Circles

The circle tool is used to create circles for cuts and bosses in a sketch. The circle is defined by either **Center** or **Perimeter** creation. Center requires two locations: the center, and a location on its circumference. Perimeter requires locations that represent two (or optionally three) locations on the perimeter.

Where to Find It

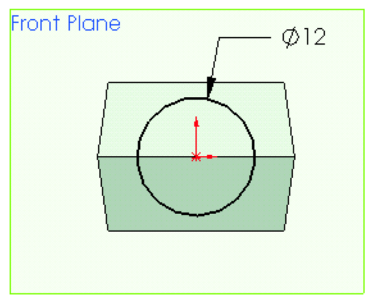
- From the **Tools** menu, select **Sketch Entities, Circle** or **Perimeter Circle**.
- Or, on the Sketch toolbar click **Circle** or **Perimeter Circle**.

Sketching the Circle

Many inference points can be used to locate circles. You can use the center of previously created circles, the origin and other point locations to locate the circle's center. In this example, we will automatically capture a coincident relation to the origin by sketching the center of the circle on it.

17 Add a circle and dimension it.
Using the **Circle Tool**, add the circle at the origin.

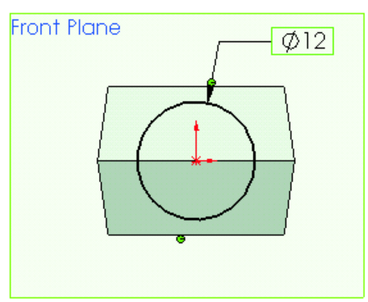
Add the diameter dimension to fully define the sketch. Set the value to be **12mm**. The sketch is fully defined.



Changing the Appearance of Dimensions

With the dimensioning standard currently in use, diameter dimensions are displayed with one arrow outside the circle. You can change the display so that two arrows are inside of the circle.

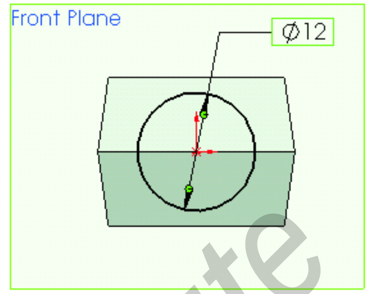
18 Click the dimension.
Two small green dots will appear on the arrowheads of the dimension.



19 Toggle the arrows.

Click one of the green dots to toggle the arrows to the inside of the circle. This works on all dimensions, not just diameter dimensions.

Click again to place the arrows outside.



20 Hide the `Front Plane`.

21 Change to `Isometric` view.

Unlike when you created the first feature, the system will not switch view orientations automatically for any other bosses or cuts. Use the **View Orientation** dialog box or the Standard Views toolbar to change to an `Isometric` view.

Extruding Up To Next

The sketch will be extruded up to the next face(s) it encounters along its path. It is important to watch the preview graphics to determine that the boss is going in the proper direction, reversing the direction if necessary.

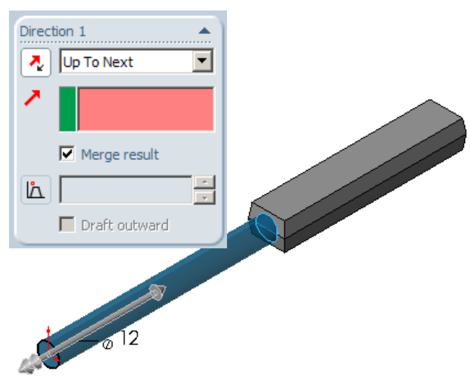
22 Up To Next extrusion.

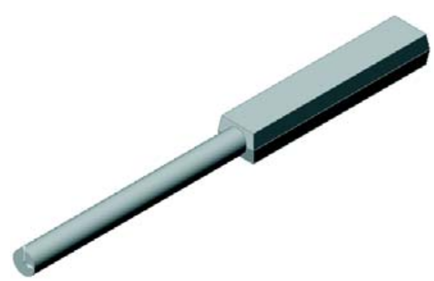
Click **Insert, Boss/Base, Extrude...** and watch the preview display. Change the direction so that the preview shows the extrusion running towards the `Handle`.

Change the end condition to **Up To Next**.

Click **OK**.

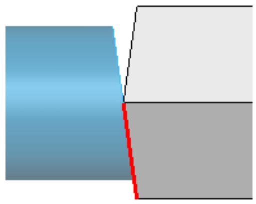
Rename the feature to `Transition`.

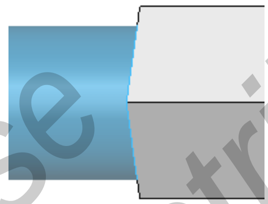




Up To Next vs. Up To Surface

The end conditions **Up To Next** and **Up To Surface** generate different results in many cases. The image on the left is for Up To Surface when the angled (red) face is selected. The extrusion is shaped by the selected surface. Only one surface selection is allowed. The image on the right is for Up To Next. All faces in the path of the extrusion are used to shape the extrusion.



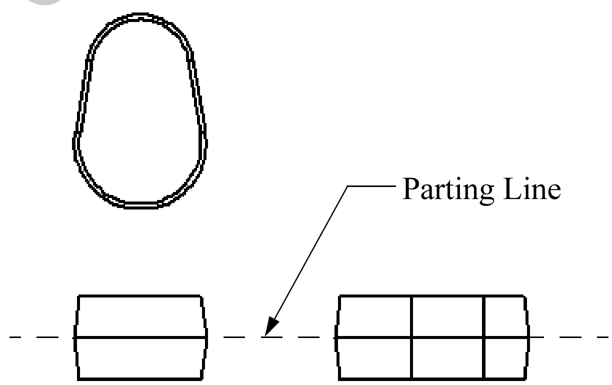


Design Intent of the Head

The `Head` is a sketched feature that uses lines and tangent arcs to form the basic outline or profile. The profile is extruded in opposite directions, equally, with draft. This feature is the key feature of the part. It will contain pockets and holes used for the location of other parts.

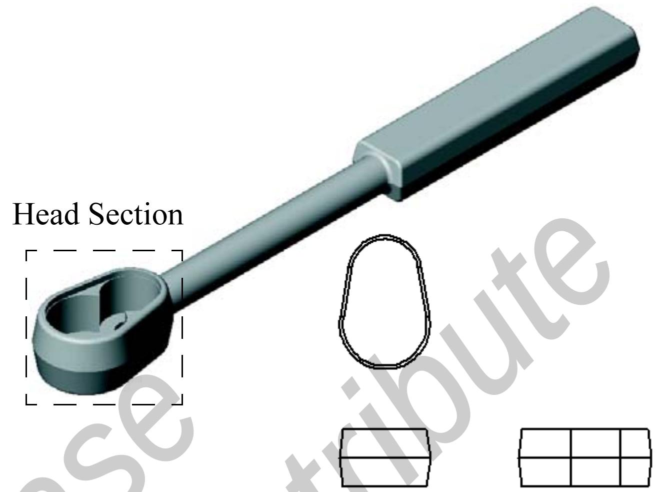
The design intent of the `Head` is listed below:

- **Arc Centers:** The centers of the two arcs in the outline (profile) line up vertically in a `Top` view orientation. The radii are not equal, and can change to any value.

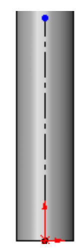


- **Profile Location:** The sketch geometry is located on the parting plane of the solid with the larger arc centered with respect to the model origin.
- **Draft:** The applied draft is equal on both sides of the parting plane.
- **Thickness:** The thickness of the part is equal on both sides of the parting line.

- **Symmetry:**
  The geometry is symmetrical.



23 Centerline.
Select the reference plane `Top` as the sketch plane. Orient the view to the same direction. Start off the sketch with a centerline as shown.



Introducing: Centerpoint Arc

Centerpoint Arc creates an arc based on a center, start point and an end point.

Where to Find It

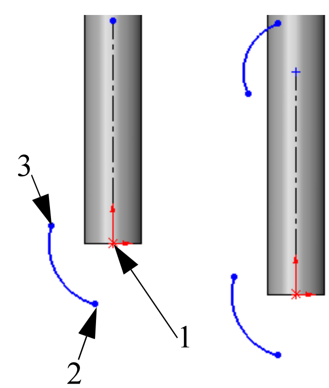
- From the **Tools** menu choose **Sketch Entities, Centerpoint Arc...**.
- Or, right-click in the graphics window and select **Centerpoint Arc**.
- Or, on the Sketch toolbar, pick the **Centerpoint Arc** tool.

24 Sketch a centerpoint arc.
Select the **Centerpoint Arc** tool and click first at the origin (1).

Move out to establish the radius and start point and click again (2).

Move to establish the end point and click a final time (3).

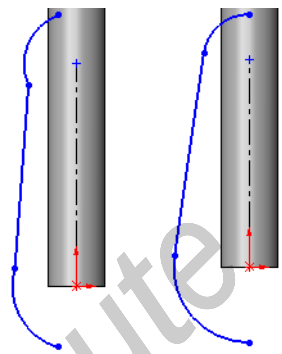
Repeat the procedure for an arc on the open end of the centerline.



25 Complete the sketching.

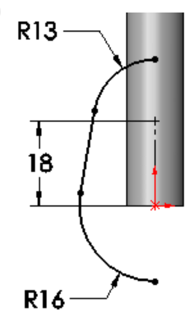
Sketch a line from endpoint to endpoint. Note that the endpoints are not tangent to the line. Add these relations:

- **Tangent** between the line and each arc.
- **Coincident** between the centerline and each open arc endpoint.



26 Add dimensions.

Add one linear and two radial dimensions to the sketch.



Introducing: Mirror Entities

Mirroring requires a line, linear edge or centerline. This line defines the mirror plane which is always normal to the sketch plane and passes through the selected centerline.

Where to Find It

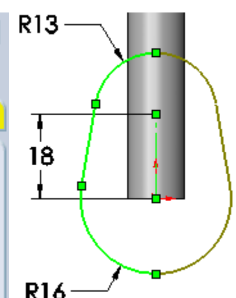
- From the **Tools** menu choose: **Sketch Tools, Mirror**.
- Or, on the Sketch toolbar click **Mirror Entities**.

27 Mirror.

Click the **Mirror** tool. Select the two arcs and connecting line as **Entities to mirror**.

Click in **Mirror about** and select the centerline.

The geometry will be mirrored about the centerline.



Driven Dimensions

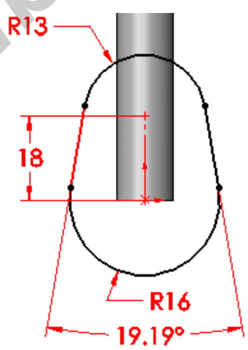
Driven or **Reference Dimensions** can be created in any sketch. SolidWorks guides you towards creating this type whenever dimensions are added to geometry that is already fully defined. A **Driven** dimension is indicated with a color difference. The driven dimension will always display the proper value but can never be used to force a change in the model.

Overdefined Sketches

If the status of the sketch changes to from fully defined to over defined (see *Status of a Sketch* on page 29), a diagnostic tool appears. This tool can be used to repair the sketch. Other unfavorable states can be repaired as well.

---

28 Angular dimension.
Click the **Dimension** tool and select the pair of angled lines. Position the dimension text below the sketch, between the lines.



29 Driven message.
The next message gives you the choice of making the dimension driving or driven. The default selection, **Make this dimension driven**, is controlled by **Tools, Options**. Select **Leave this dimension driving** and click **OK**. The sketch becomes **Over Defined**.

Resolve Conflicts

The **Resolve Conflicts** option is used to repair Over Defined, No Solution Found, or Invalid Solution Found conditions in the sketch.

Note

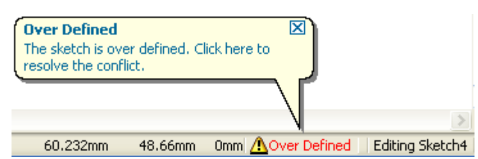
General part editing and repairs will be discussed in *Lesson 7: Editing: Repairs*.

Where to Find It

- Click the **Over Defined** (or other condition) button in the lower right corner.

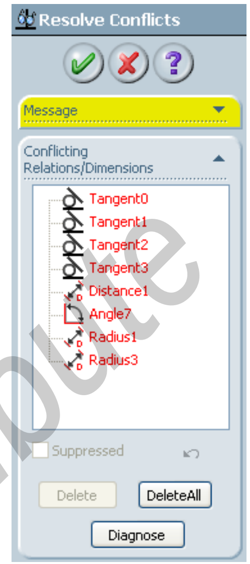
---

30 Over defined.
When the sketch becomes Over Defined, a message pops up from the lower right corner of the screen. Click the **Over Defined** button.



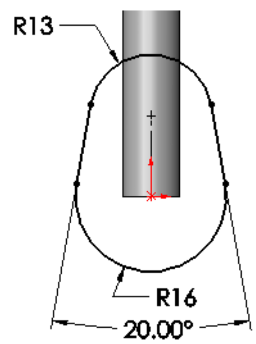
31 Diagnose.
Click **Diagnose** to determine possible solution sets to resolve the over defined state.

Deleting and of these sets will remove the over defined state.

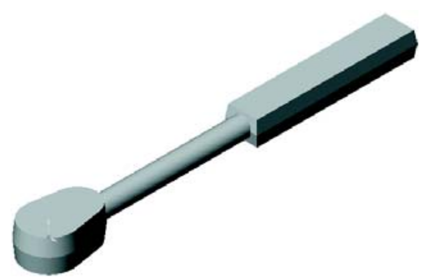


32 Delete.
Select the `Distance1` relation and click **Delete**.

33 Conflict resolved.
Set the angle dimension to **20** degrees.



34 The extrusion.
Change to an `Isometric` view and click **Insert, Boss/Base, Extrude...** from the menu. Set the type to **Mid Plane**, depth to **20mm** and draft to **6°**. Rename the latest feature to `Head`.



The three main features that make up the overall shape of the part are now complete.

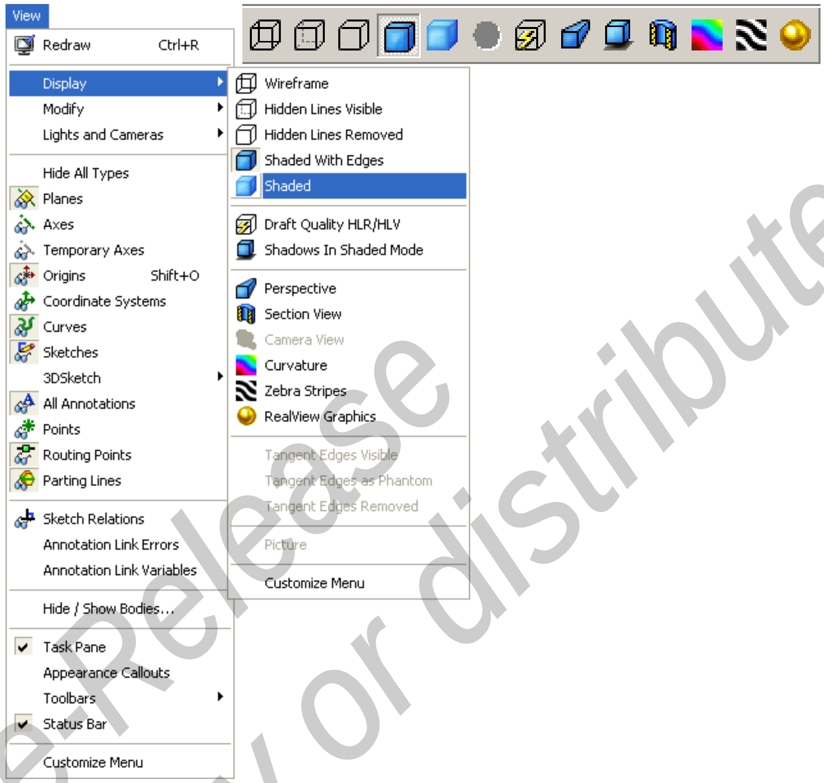
View Options

The SolidWorks software provides you with many options for controlling and manipulating how models are displayed on your screen. In general, these view options can be divided into two groups. These groups correspond to the two sub-menus that are available on the **View** menu and the two groups of tools on the view toolbar.

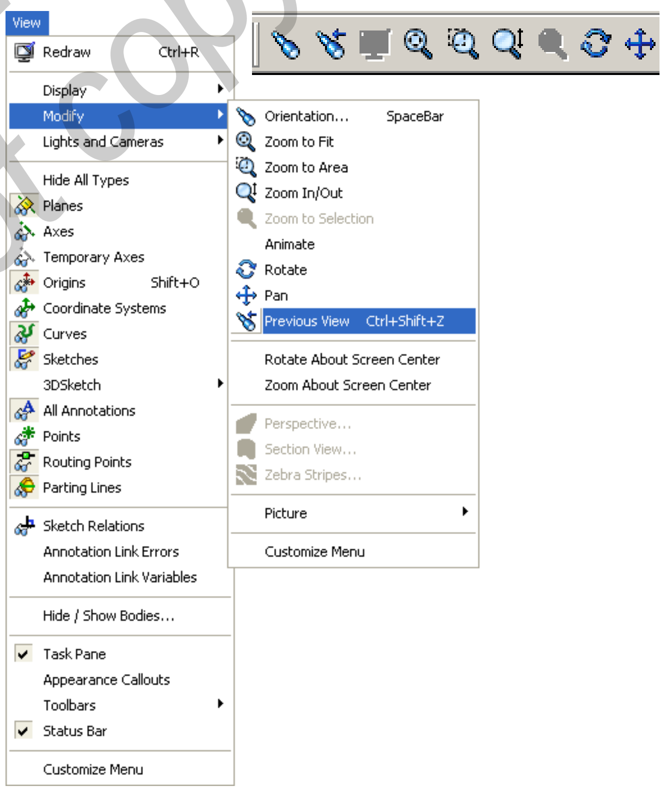
Note

These view options are available for use in single and multiple viewport situations. For more information, see *Viewports* on page 63.

Display Options

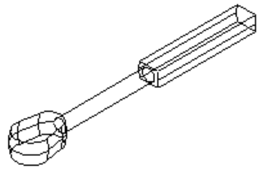


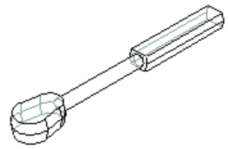
Modify Options

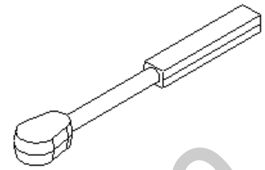


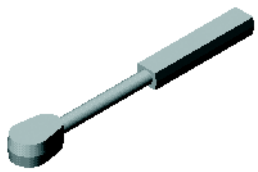
Display Options

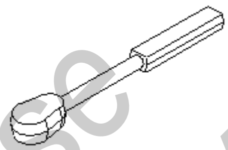
The following illustrations of the `Ratchet` illustrate the different types of display options.

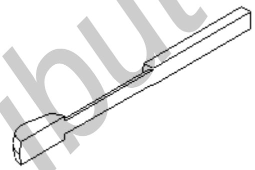

Wireframe


Hidden Lines Visible


Hidden Lines Removed

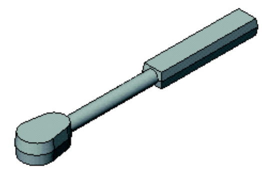

Shaded


Perspective


Section


Zebra Stripes


Shadows in Shaded Mode


Shaded With Edges

Note

The **Perspective** and **Section** view options can be applied to any type of view – wireframe, hidden line, or shaded. The **Draft Quality HLR/HLV** tool can be active with all view types but affects only the **Hidden Lines Removed** and **Hidden Lines Visible** options by making the display faceted and faster to manipulate.

Modify Options

The modify options are listed below next to their corresponding tools. Your instructor will demonstrate these during class.

Note

It is notoriously difficult to illustrate something as dynamic as view rotation via a medium as static as a printed manual. Therefore, the different view options are only listed and summarized here. Your instructor will demonstrate them for you in class.


Zoom to Fit: Zooms in or out so the entire model is visible.


Zoom to Area: Zooms in on a portion of the view that you select by dragging a bounding box. the center of the box is marked with a plus (+) sign.


Zoom In/Out: Zooms in as you press and hold the left mouse button and drag the mouse up. Zooms out as you drag the mouse down.


Zoom to Selection: Zooms to the size of a selected entity.

Rotate View: Rotates the view as you press and hold the left mouse button and drag the mouse around the screen.

Pan View: Scrolls the view so the model moves as you drag the mouse.

Middle Mouse Button Functions

The middle mouse button on a three button mouse can be used to dynamically manipulate the display. Using the middle mouse button you can:

- **Rotate the view**
  Press and hold the middle mouse button. As you move the mouse, the view rotates freely.

Tip

To rotate *about* a vertex, edge, axis or temporary axis:

Click the middle mouse button on the geometry. As you move the mouse, the view rotates about that selected geometry.

- **Pan or scroll the view**
  Press and hold the **Ctrl** key together with the middle mouse button. The view will scroll as you drag the mouse.
- **Zoom the view**
  Press and hold the **Shift** key together with the middle mouse button. The view will zoom larger as you drag the mouse upward; smaller as you drag the mouse downward.

Note

In a drawing, only the **Zoom** and **Pan** functions can be used.

Keyboard Shortcuts

Listed below are the predefined keyboard shortcuts for view options:

- **Arrow Keys** . . . . Rotate the view
- **Shift+Arrow Keys** . . . . Rotate the view in 90° increments
- **Alt+Left or Right Arrow Keys** . Rotate about normal to the screen
- **Ctrl+Arrow Keys** . . . . Move the view
- **Shift+z** . . . . Zoom In
- **z**. . . . . Zoom Out
- **f** . . . . Zoom to Fit
- **Ctrl+1** . . . . Front Orientation
- **Ctrl+2** . . . . Back Orientation
- **Ctrl+3** . . . . Left Orientation
- **Ctrl+4** . . . . Right Orientation
- **Ctrl+5** . . . . Top Orientation
- **Ctrl+6** . . . . Bottom Orientation
- **Ctrl+7** . . . . Isometric Orientation

- **Ctrl+8** . . . . View Normal To
- **Spacebar** . . . . View Orientation dialog

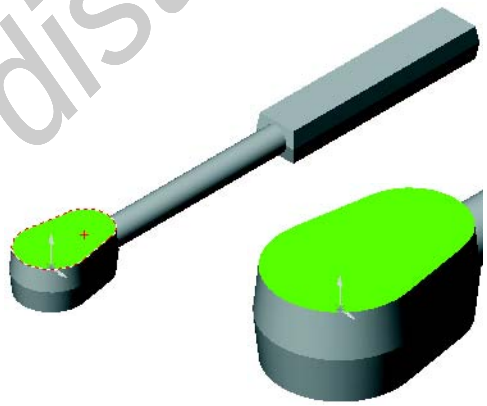
Using Model Edges in a Sketch

The first `Cut` feature to be added is the `Recess`, a pocket that is extruded down from the top face of the `Head`. This feature allows for the placement of a cover plate over the ratchet gears. Since the cover is the same general shape as the top face, it would be helpful to take advantage of the edges of the `Head` when sketching the profile for the `Recess` cut. We will do this by making an **Offset** of the edges of the `Head`.

Zoom to Selection

The **Zoom to Selection** option zooms in on a selected entity, making it fill the screen.

35 Select face and zoom.
Select the top face of the `Head` and click **Zoom to Selection**. That face will fill the graphics window.



Sketching an Offset

Offsets in a sketch rely on existing model edges or sketch entities in another sketch. In this example we will utilize the model edges of the `Head`. These edges can be chosen singly, or as the boundary of an entire face.When possible, it is a good idea to pick the face because the sketch will regenerate better if subsequent changes add or remove edges from the face.

The edges are projected onto the plane of the sketch, regardless whether they lie on that plane or not.

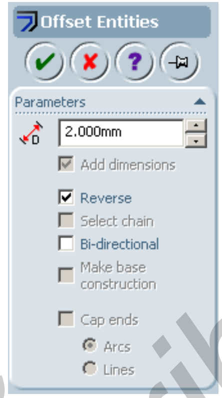
Introducing: Offset Entities

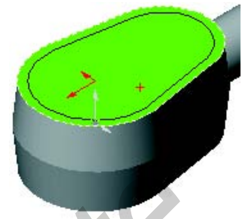
Offset Entities is used to create copies of model edges in a sketch. These copies are offset from the original by some specified amount.

Where to Find It

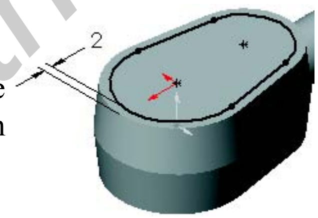
- From the **Tools** menu, select **Sketch Tools, Offset Entities...**.
- Or, on the Sketch toolbar click **Offset Entities**.

36 Offset the face boundaries.
Select the top face and click the **Sketch** tool. With the face still selected click the **Offset** tool on the toolbar. Set the distance value to **2mm** and **Reverse** the direction if necessary, moving the offset to the inside.

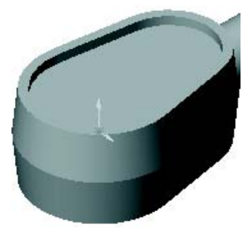




37 Resulting Offset.
The offset creates two lines and two arcs. This geometry is dependent on the solid face it came from and will change with the solid. The sketch is automatically fully defined and ready to extrude as a cut.



38 Settings for the cut.
Choose a **Blind** cut with **2mm** for the depth value and click **OK**.

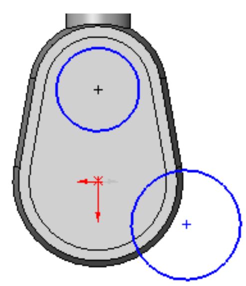


39 Rename the feature.
Change the name of the feature to `Recess`.

Creating Trimmed Sketch Geometry

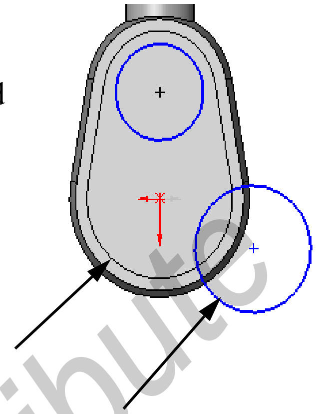
The `Pocket` is another cut feature, applied to a planar face of the model. This sketch uses overlapping circles that are trimmed to create a single contour. The centers of the circles are related to existing circular centerpoints.

40 Sketch circles.
Select the top, inner face created by the last feature as the sketch plane. Using the **Circle** tool, create a circle using the existing centerpoint location as the circle's origin. Snapping to this location will relate the circle to it automatically. Create a second circle off to the side of the model.



41 Relate the centers.

Click **Add Relation** to open the **Add Relations** PropertyManager. Select the second circle and the edge of the cut. Choose the **Concentric** option and click **OK**. **Concentric** forces the two arcs (the circle and the circular edge) to share a common center. This will pull the circle into position.



Trim and Extend

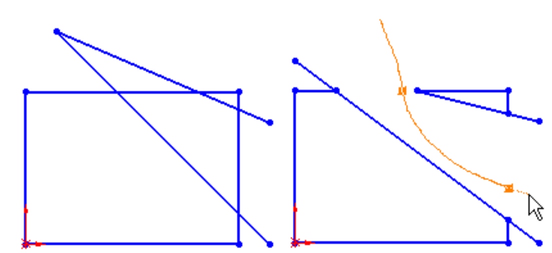
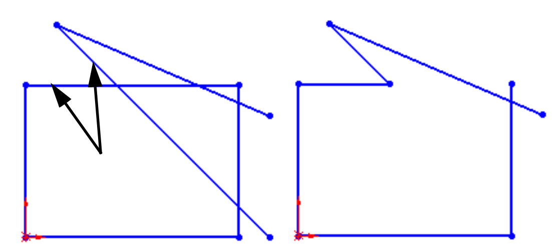
Sketch entities can be trimmed shorter using the **Trim** option. In this example, the overlapping portions of the circles will be removed. There are several trimming options: **Power Trim**, **Corner**, **Trim away inside**, **Trim away outside** and **Trim to closest**. They can also be lengthened using **Extend**. They are discussed below.

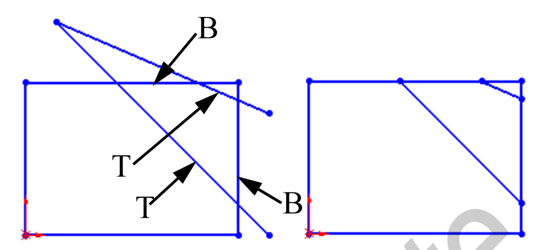
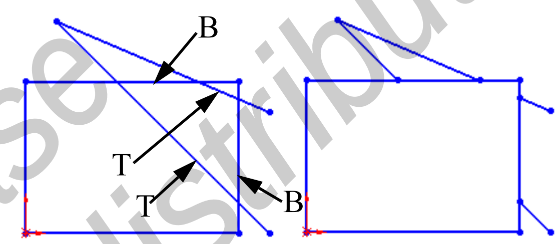
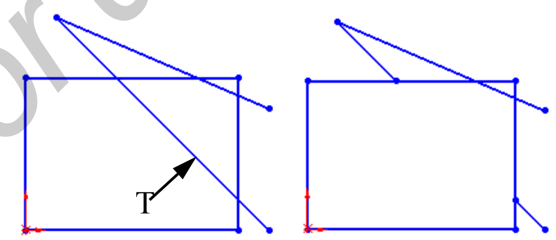
Introducing: Trim

Trim can be used to shorten sketch geometry.

Where to Find It

- From the **Tools** menu, select **Sketch Tools, Trim**.
- Or, on the Sketch toolbar click **Trim Entities** .

Power trim removes the portion of an entity that you drag over between intersections or to an endpoint.	
The **Corner** option is used to trim by keeping the geometry selected to a common intersection.	

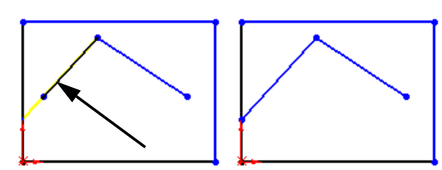
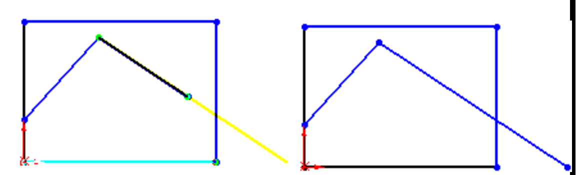
Use the **Trim away outside** to keep the *inside* portions if geometry against a boundary. Select the two boundaries (B) first then the pieces of geometry to trim (T).	
Use **Trim away inside** to keep the *outside* portions if geometry against a boundary. Select the two boundaries (B) first then the pieces of geometry to trim (T).	
Use **Trim to closest** to trim selected geometry to the nearest intersection or remove a portion of the geometry between boundaries.	

Introducing: Extend

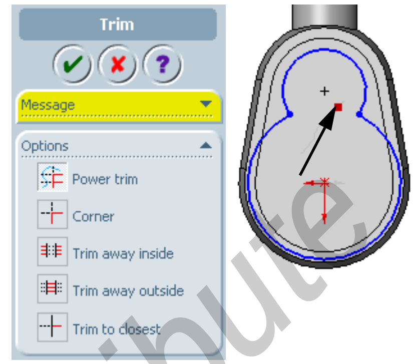
Extend can be used to lengthen sketch geometry.

Where to Find It

- From the **Tools** menu, select **Sketch Tools, Extend**.
- Or, on the Sketch toolbar click **Extend Entities**.

Select nearest the end to extend and click to extend to the next intersection.	
Drag the nearest endpoint and drop it on the intersecting entity to extend.	

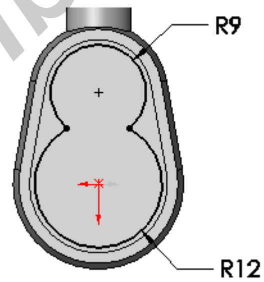
42 Trim the circles.
Click on the **Trim** tool and select the **Power trim** option.



Rule *Drag across the portions of the sketch entities that you want to remove.*

The system will find the intersections between the circles and remove the excess.

43 Add dimensions.
Add dimensions to the arcs. This will fully define the sketch.

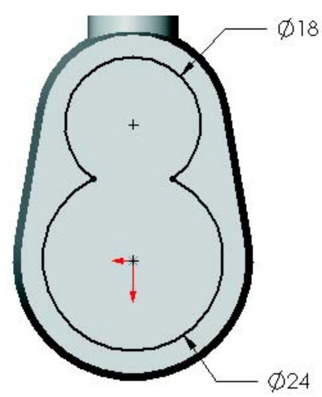


44 Turn off the dimension tool.
An easy way to turn off the dimension tool is to simply press the **Esc** key on the keyboard.

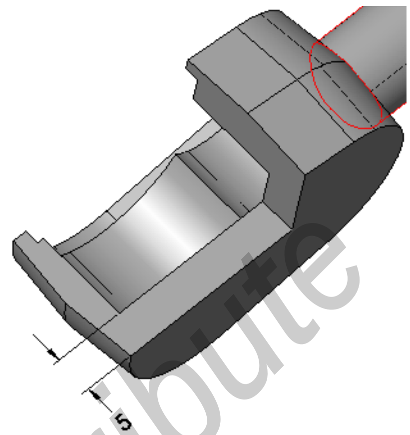
Modifying Dimensions

Since the sketch entities are arcs, the system automatically created radial dimensions. If you prefer diameter dimensions, you can quickly change the display options. For more in-depth dimension changes, right-click the dimension, and select **Properties**.

45 Diameter dimensions.
Select the dimensions, right-click and choose **Display Options, Display As Diameter**.



Introducing: Offset From Surface

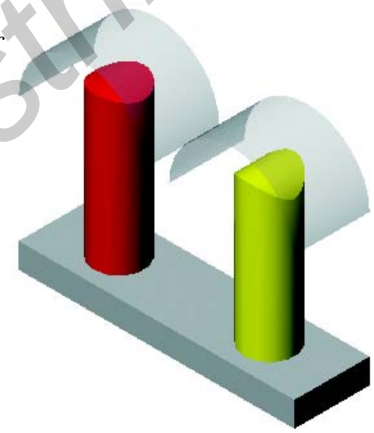


The **Offset From Surface** end condition is used to locate the end of an extrusion as a measurement from a plane, face or surface rather than the sketch plane of the feature.

In this example the end of the extrusion is measured from the bottom face of the part.

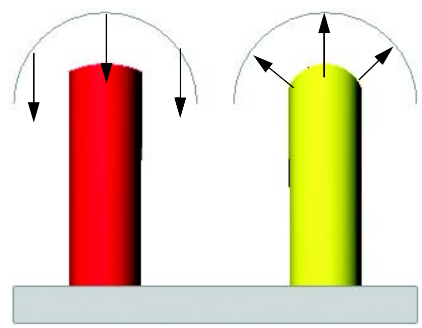
The **Translate Surface** option can be checked or cleared. Its meaning is explained below.

What the Translate Surface option does:



The **Translate Surface** option of the **Offset From Surface** end condition is *off* by default.

In the illustration at the right, both columns are positioned below two identical semi-circular reference surfaces. Both columns are extruded such that the top of each is **1.4”** below the reference surfaces. The column on the *left* was extruded with the **Translate Surface** option on. The column on the *right* was extruded with the option off.



The **Offset from Surface** option in the **Translate Surface** option defines the end condition by linearly translating a copy of the surface in the direction of the extrusion. Without it, the copied surface is created by projection normal to the original surface. Hence the two different results.

Note

In this example, the position of the planar face selected means that both options reach the same result.

---

46 Offset From Surface.
Click the **Extruded Cut** icon and choose the **Offset From Surface** end condition. Set the **Offset Distance** to **5mm**.

Introducing: Select Other

Select Other is used to select hidden faces of the model without reorienting it.

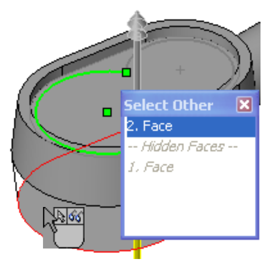
Select Other Procedure

To select faces that are hidden or obscured, you use the **Select Other** option. When you position the cursor in the area of a face and press the right mouse button, **Select Other** is available as an option on the shortcut menu. The face closest to the cursor is hidden and listed as **1.** in the dialog under --*Hidden Faces*--. Other visible faces are numbered and listed in the dialog. Moving over them in the dialog highlights them on the screen.

The reason the system hides the closest face is since that one was visible, if you wanted to select it you would have simply picked it with the left mouse button.

47 Face selection.

Right-click over the hidden bottom face and choose **Select Other**. Slide the cursor up and down the Select Other list to highlight possible face selections. Use the left mouse button to select the face directly or select the choice **2. Face** from the list.



Rename the feature `Pocket`.

Tip

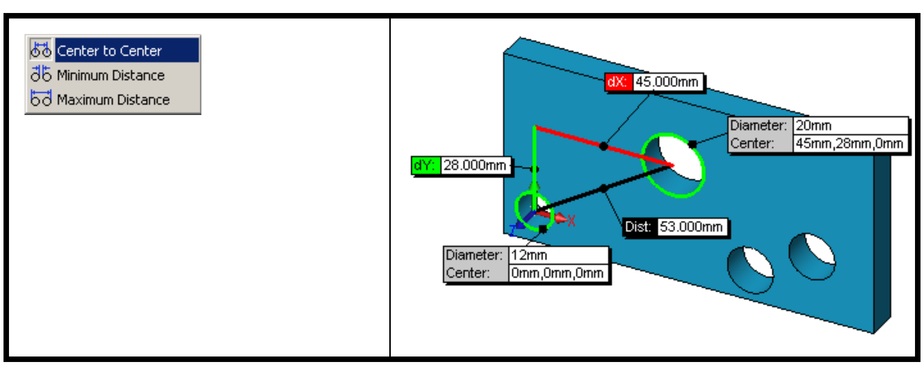
Other faces can be added to the --*Hidden Faces*-- list. Right-click a face to hide it. Press **Shift** and right-click to unhide it and remove it from the list.

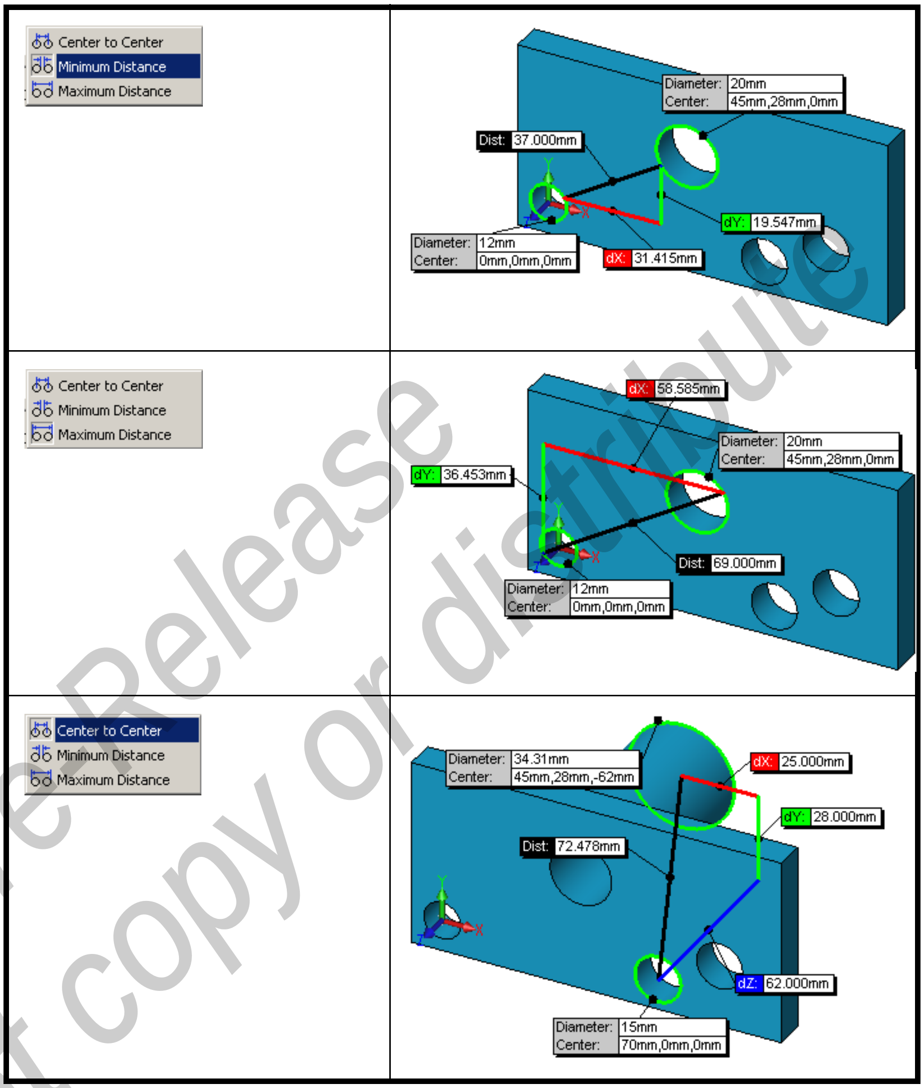
Measuring

The **Measure** option can be used for many measurement tasks. Here it is used to measure the shortest distance between an edge and a plane. It can measure geometry including vertices, edges and faces.

Introducing: Measure

The **Measure** command can calculate distances, lengths, surface areas, angles, circles and X, Y, Z locations of selected vertices. For circles and arcs, the center, minimum and maximum dimensions are available as shown below.





Where to Find It

- On the Tools toolbar, click the **Measure** tool.
- Or from the **Tools** menu choose **Measure...**.

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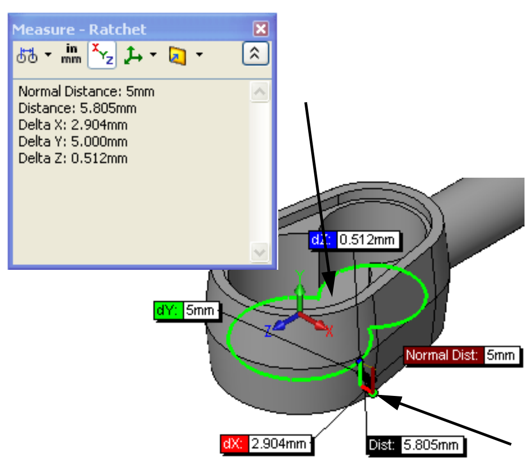
48 Measure between face and vertex.

Click the **Measure** tool and select the face and vertex shown.

The **Normal Distance** is **5mm**. Information for the combined selections is displayed.



Tip The **Status Bar** at the bottom of the SolidWorks window displays some similar information when the Measure tool is off. If a circular edge was selected, the status bar would show the **Radius** and **Center**.

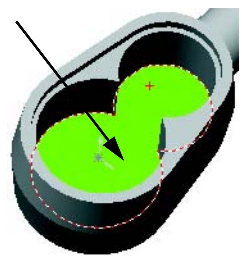
Using Copy and Paste

The `Ratchet` requires two through holes of different diameters. We will create one hole and copy and paste it to make the second.

Sketching the Hole

Circular holes are very simple to create. A sketch circle, related to the model and dimensioned, is all you need. The Hole Wizard could also be used to create this hole.

49 Open a sketch.
Click on the inner bottom “figure eight” face and open a new sketch.



50 Create a circular hole.
Sketch a circle centered on the upper center mark and add a dimension. Set the diameter to **9mm** and create a **Through All** cut.

Name the feature `Wheel Hole`.

Copy and Paste Features

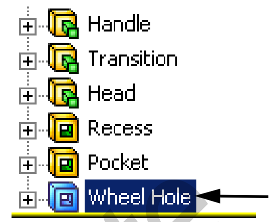
Simple sketched features and some applied features can be copied and then pasted onto a planar face. Multi-sketch features such as sweeps and lofts cannot be copied. Likewise, certain applied features such as draft cannot be copied, although fillets and chamfers can.

Once pasted, the copy has no ties or associativity to the original. Both the feature and its sketch can be changed independently.

Copying a Feature

Copy features by selecting them and using the standard Windows shortcut **Ctrl+C** or picking the **Copy** tool on the Standard toolbar. You can also select **Copy** from the **Edit** menu. Finally, you can employ the standard Windows “drag and drop” technique while holding down the **Ctrl** key.

51 Identify the feature to copy.

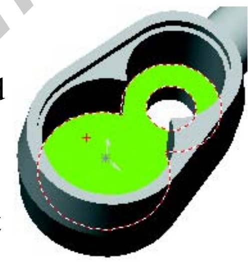


The feature to be copied must be identified either in the FeatureManager design tree or on the model. For this example, select the feature `Wheel Hole` by picking it in the FeatureManager design tree. Next, copy it to the clipboard using the **Copy** option on the Standard toolbar.

Note

You may also use **Ctrl+C** or **Edit, Copy** to create a copy on the clipboard.

52 Select the face on which to paste.



The copied feature must be pasted onto a *planar* face. Select the bottom inner face, the same one used for the sketch plane of the `Wheel Hole`.

53 Paste the feature.

Paste the copy using the **Paste** tool, the shortcut **Ctrl+V**, or **Edit, Paste**.

54 Copy confirmation.

The `Wheel Hole` was **Concentric** to the smaller end of the "figure eight" face. The copy carries that **Concentric** relation with it, except the system now has a bit of a problem. It doesn't know what edge to make it concentric to. Therefore, we are given three choices:

- Delete the relationship.
- Keep it even though it is unresolved (dangling).
- Cancel the copy operation altogether.

55 Click Delete.

Dangling Relations

Dimensions and relations are said to be dangling when they reference something that has been deleted or that is otherwise unresolved. Dangling relations can usually be repaired through one or more techniques. We will discuss repairing dangling relations later in the course in *Lesson 7: Editing: Repairs*.

56 Feature pasted.

The feature and its sketch are added to the FeatureManager design tree and the model. Note that the feature is not centered. That is because its sketch is, in fact, under defined.

57 Find the sketch.
Click the sign preceding the pasted feature in the FeatureManager design tree.

Editing a Sketch

Once created, sketches can be changed using **Edit Sketch**. This opens the selected sketch so that you can change anything: the dimension values, the dimensions themselves, the geometry or geometric relations.

Introducing: Edit Sketch

Edit Sketch allows you to access a sketch and make changes to any aspect of it. During editing, the model is "rolled back" to its state at the time the sketch was created. The model will be rebuilt when the sketch is exited.

Where to Find It

- From the **Edit** menu, choose **Sketch**.
- Or, right-click the feature whose sketch you want to edit and select **Edit Sketch**.

Relate and Change the Sketch

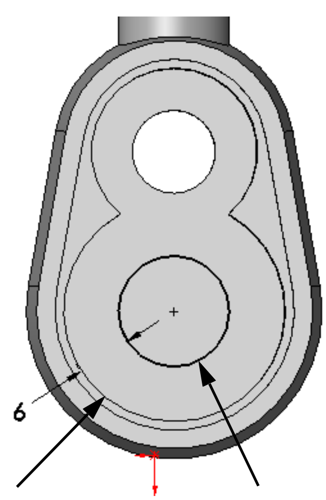
Since the copy has no relations to the model geometry or the origin, the sketch is under defined and should be brought up to a fully defined state. Use geometric relations to do this.

58 Edit the sketch of the copied feature.
The copied feature includes both the feature itself and its sketch. The sketch defines the shape and size of the profile as well as the location. Right-click the feature or its sketch, and select **Edit Sketch**.

59 Relation and dimension.
The circle and the diameter dimension are in the sketch. No other relations or dimensions exist to locate the circle. Delete the dimension.

Click **Add Relation** . Select the edge of the circle and the edge of the solid and use **Concentric**. Or, use **Coincident** to align the origin and circle centerpoint. The sketch is now fully defined.

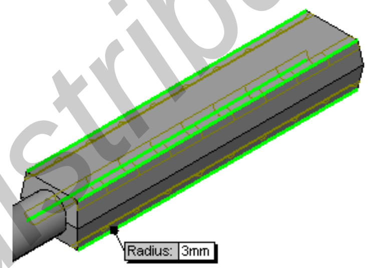
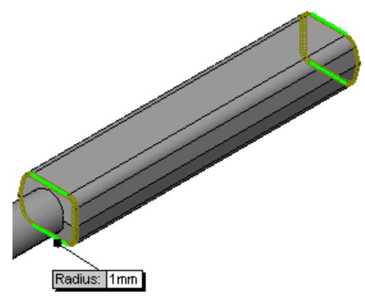
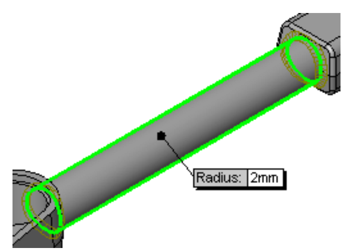
Add a **Concentric Circle Dimension** by dimensioning the circle and edge.



60 Rebuild the model.
To cause the changes to the sketch to take effect, Rebuild the model by clicking the **Rebuild** tool.

Rename the feature `Ratchet Hole`.

61 Fillets.
Add fillets on edges and faces as shown below.

R = **3mm** Name = `Handle Fillets`	
R = **1mm** Name = `H End Fillets`	
R = **2mm** Name = `T-H Fillets`	

Editing Features

The last fillet to create is around the upper and lower edges of the `Head`. Since this fillet has the same radius as the fillet on the ends of the `Handle`, we will edit this existing fillet to include the edges on the `Head`. This is a better technique than creating a new fillet and trying to figure out how to keep their radii equal. To do this we will edit the definition of the `H End Fillets`.

Introducing: Edit Feature

Edit Feature changes how a feature is applied to the model. Each feature has specific information that can be changed or added to, depending on the type of feature it is. As a general rule, the same dialog box used to create a feature is used to edit it.

Where to Find It

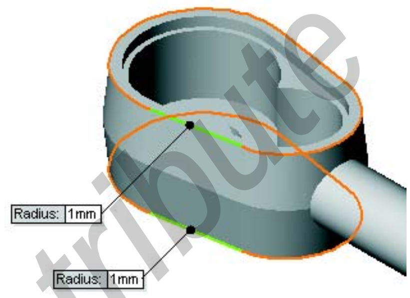
- Right-click the feature to edit – either in the FeatureManager design tree or the graphics window, and select **Edit Feature**.

Editing the Fillet

Edit the feature of the `H End Fillets` to include additional edges.

62 Select and edit the fillet.

Right-click the feature `H End Fillets`, and select **Edit Feature**. Select the additional edges around the upper and lower edges of the `Head`. The selection list should now indicate a total of 6 edges selected.

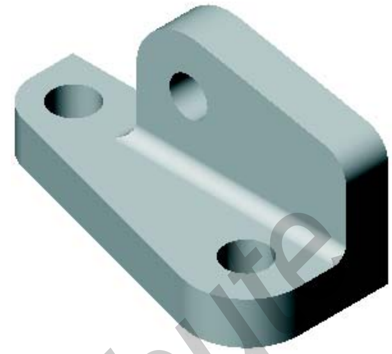


63 Save and close the part.

Exercise 10: Base Bracket

This lab reinforces the following skills:

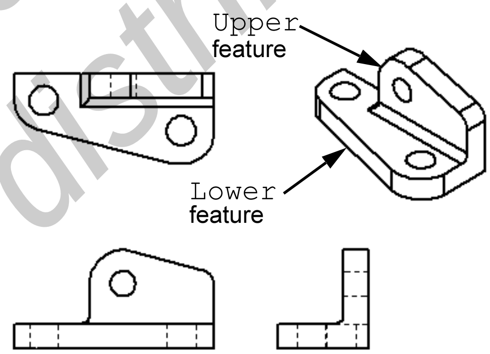
- Sketching lines.
- Adding geometric relations.
- Sketching on standard planes.
- Sketching on planar faces.
- Filleting.
- Creating cuts, holes and bosses.



Design Intent

Some aspects of the design intent for this part are:

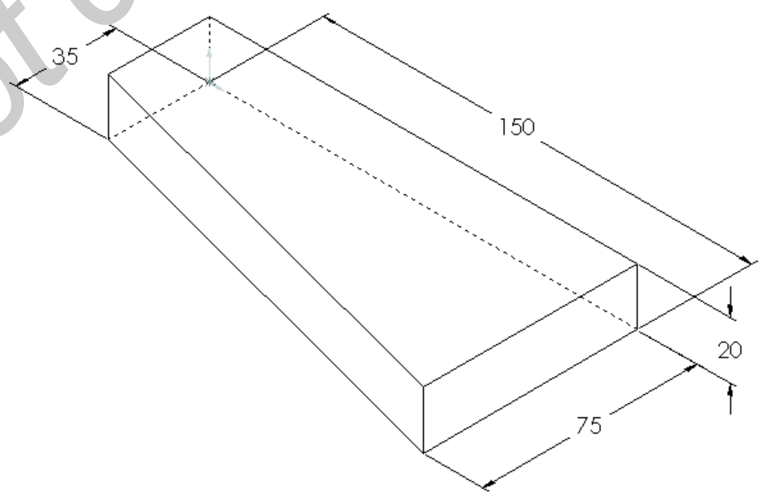
1. Thickness of the `Upper` and `Lower` features are equal.
2. The holes in the `Lower` feature are equal diameter and will remain so.
3. The `Upper` and `Lower` features are flush along the back and right side.



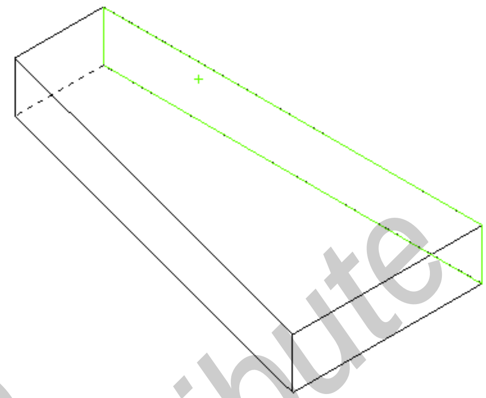
Procedure

Open a new part using the `Part_MM` template.

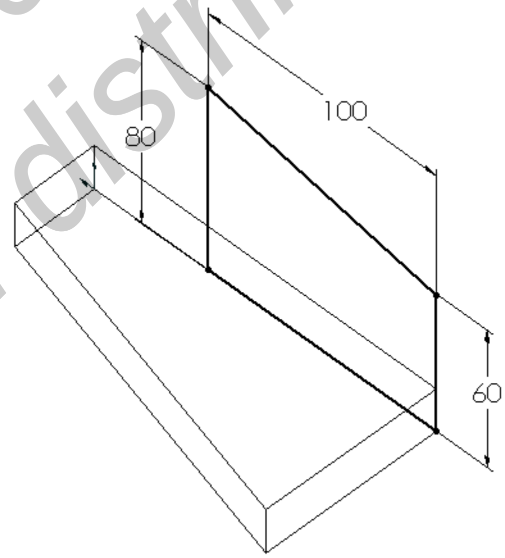
1 Create the `Lower` feature.
Use lines to sketch this profile. Add dimensions to fully define the sketch.



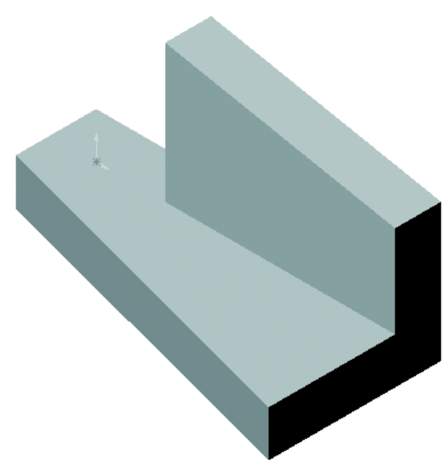
2 Select a face as sketch plane.
Select the rear face that is hidden by the top face of the model as the sketch plane. Use **Select Other** or rotate the view to select it.



3 Create the `Upper` boss feature.
Sketch the lines and relate them to the existing edges where they should be coincident.

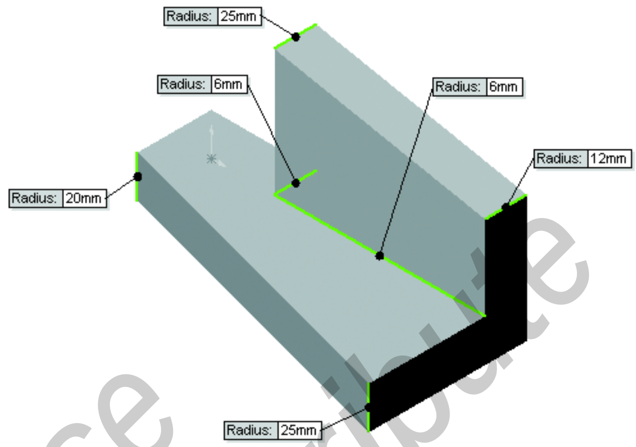


4 Extrude.
Extrude *into* the first feature a depth of **20mm.**



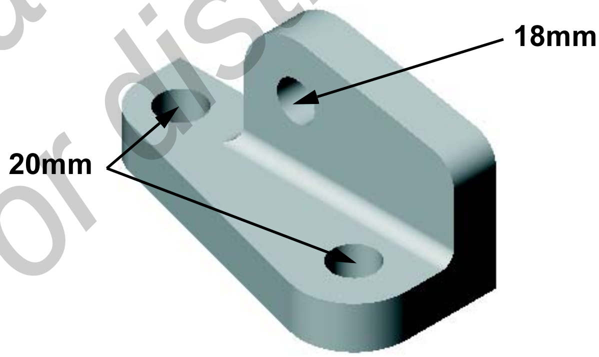
5 Create fillets and rounds.
Add the fillets in as few steps as possible.

Rename the features according to fillet size.



6 Holes.
Add the holes using as few features as possible. Make sure that the holes lie concentric to the fillet radii.

For the Hole Wizard, use ANSI Metric Drill Sizes.



7 Save and close the part.

Exercise 11: Ratchet Handle Changes

Make changes to the part created in the previous lesson.

This exercise uses the following skills:

- Editing sketches.
- Editing features.

Design Intent

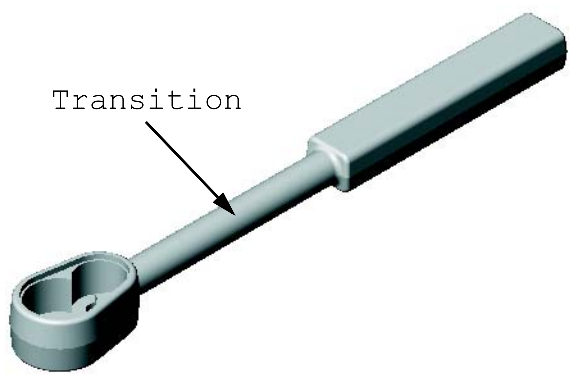
Some aspects of the design intent for this part are:

1. The part must remain symmetrical about the Right reference plane.
2. The Transition requires flats that are driven by the distance between them.

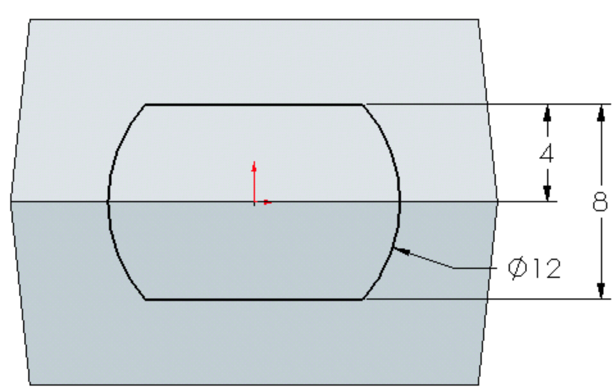
Procedure

Open an existing part.

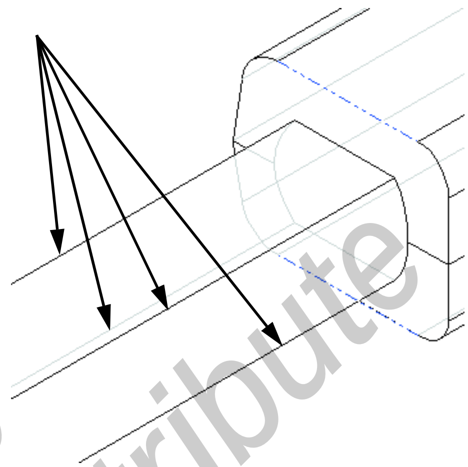
1 Open the part Ratchet Handle Changes.
The change will take place in the shape of the Transition feature.



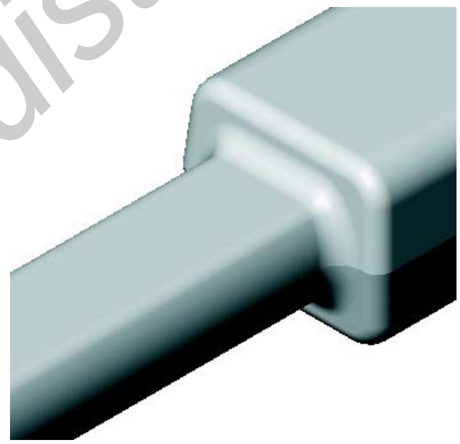
2 Edit the sketch.
Right-click the Transition feature from the screen and choose **Edit Sketch**. Modify the sketch to add the equally spaced horizontal flats **8mm** apart. Exit the sketch.



3 Edit Feature.
Edit the `H End Fillets` feature to add more edges. Select the four new edges created by the flats. Click **OK**.



4 Resulting fillets.
The new edges become part of the fillet feature, causing the shape of the next fillet feature to update.

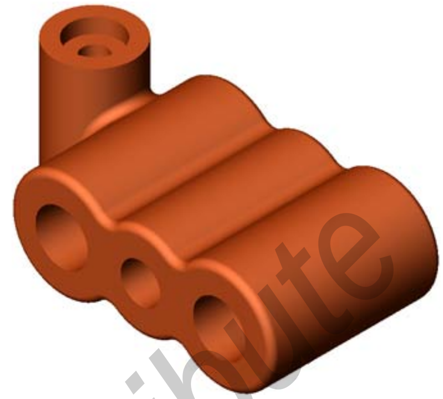


5 Save and close the part.

Exercise 12: Tool Holder

This lab reinforces the following skills:

- Sketching.
- Adding geometric relations.
- Trimming.
- Fillets.
- Creating cuts, holes and bosses.



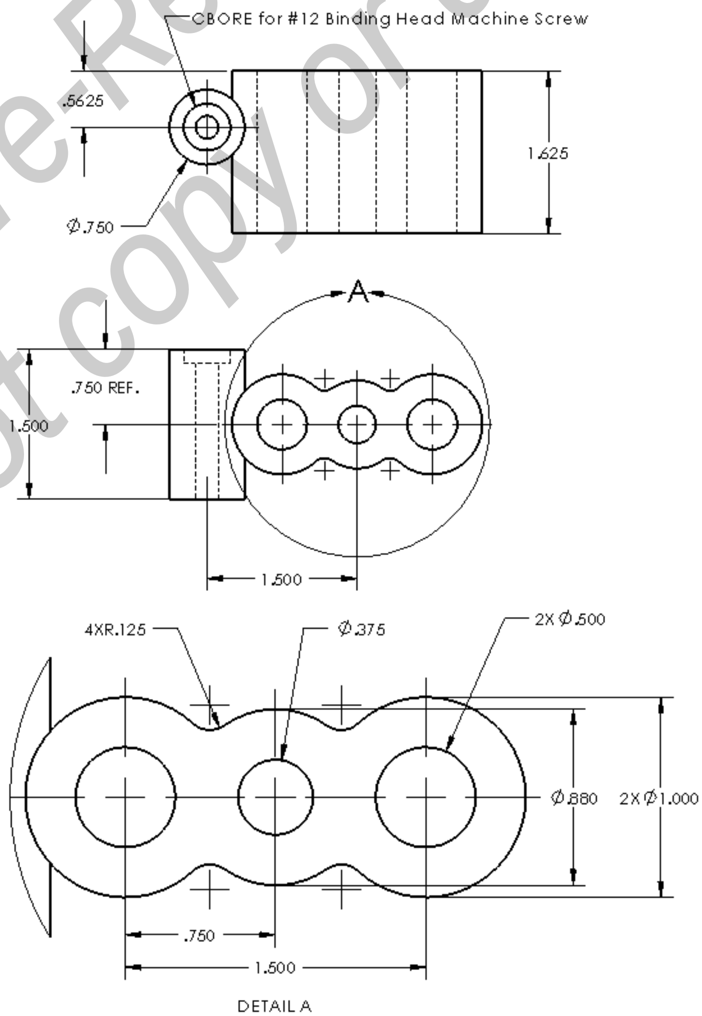
Design Intent

Some aspects of the design intent for this part are:

1. All fillets and rounds **0.0625"** unless otherwise noted.
2. Circular edges of equal radii/diameter should remain equal.

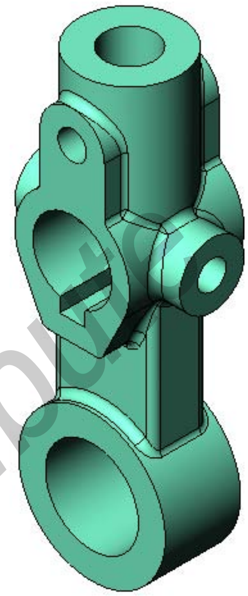
Dimensioned Views

Use the following graphics with the design intent to create the part.



Exercise 13: Idler Arm

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent. Give careful thought to the best location for the origin.



This part can be constructed using only the `Top`, `Front` and `Right` reference planes.

This lab uses the following skills:

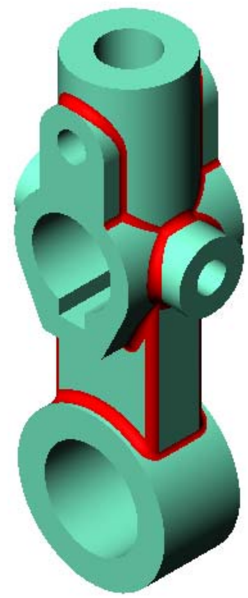
- Sketching with symmetry.
- **Mid-plane** and **Through All** extrusions
- Filleting.

Units: **inches** or **mm**

Design Intent

The design intent for this part is as follows:

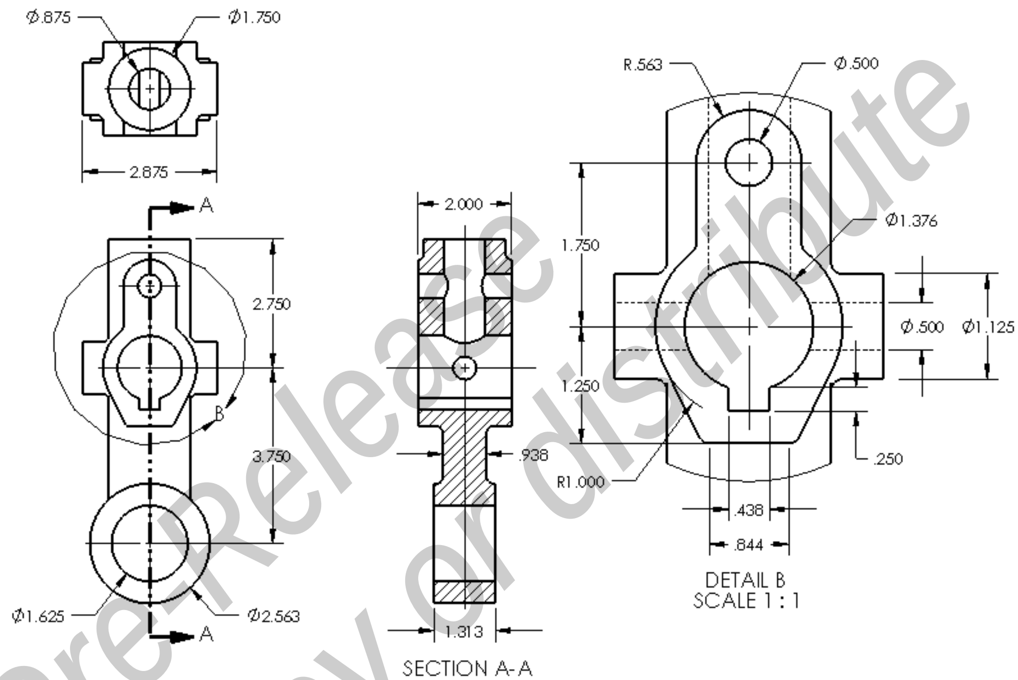
1. The part is symmetrical.
2. Front holes on centerline.
3. All fillets and rounds (highlighted red) are **R 0.125”** or **R 3mm** unless noted.
4. Center holes in `Front` and `Right` share a common centerpoint.



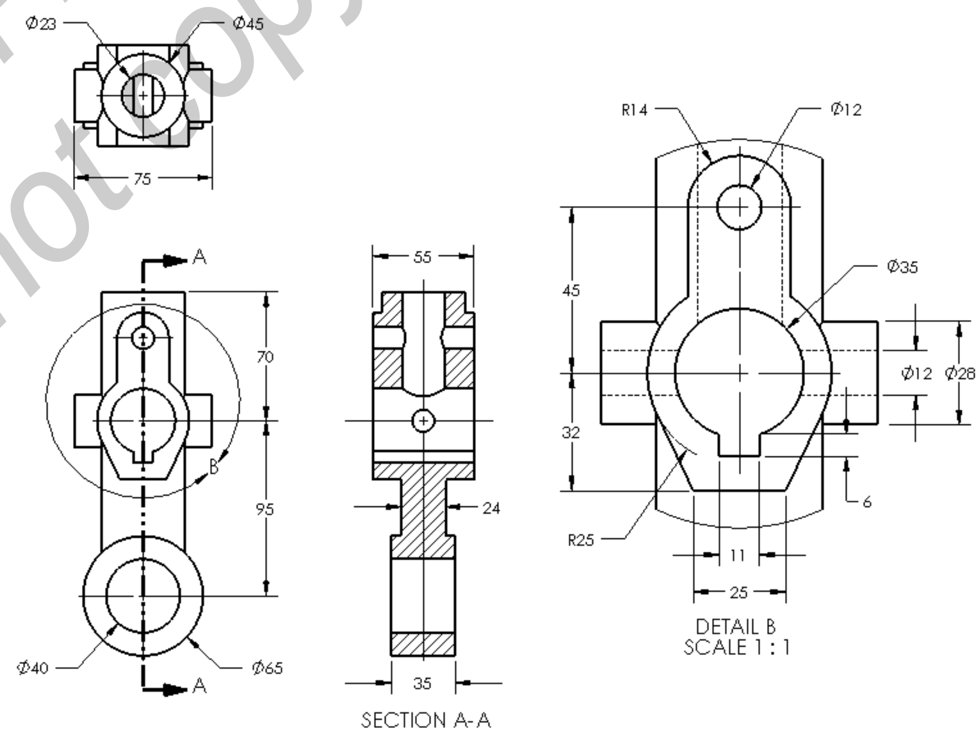
Dimensioned Views

Use the following graphics with the design intent to create the part in inches or mm. The metric values have been changed to be whole number values.

Dimensions in **inches**:



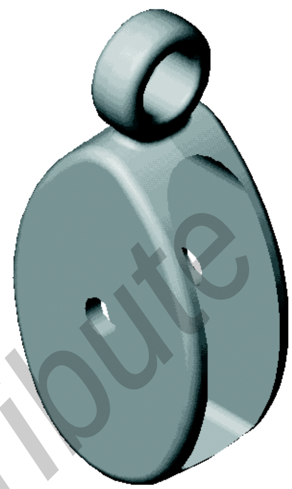
Dimensions in **mm**:



Exercise 14: Pulley

This lab reinforces the following skills:

- Creating draft while extruding.
- **Mid-plane** extrusions.
- Filleting.



Design Intent

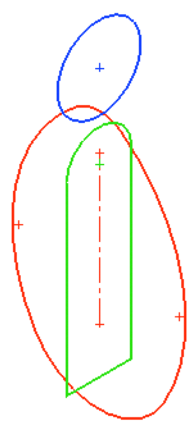
Some aspects of the design intent for this part are:

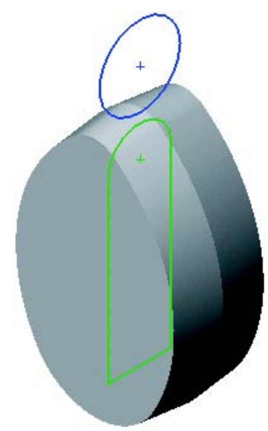
1. All fillets are **1mm** unless noted.
2. Draft is **6°** on both body and hanger.

Procedure

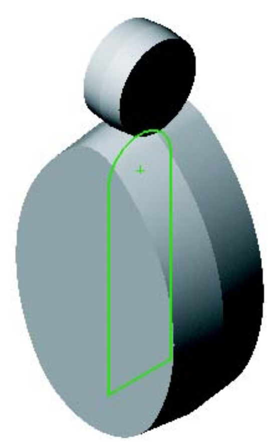
Open an existing part named `Pulley`.

1 Extrusion with draft.
Extrude the `Base` sketch (red) **10mm** using the **Mid-plane** end condition and **6°** of draft.





2 Hanger.
Use the `Hanger` (blue) sketch and another **Mid-plane** extrusion of **4mm** with the same amount of draft.

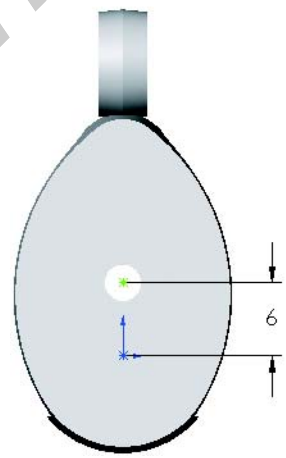


3 Cut and hole.
Create a cut using the `Center Cut` sketch (green). The cut is **Through All** in both directions.

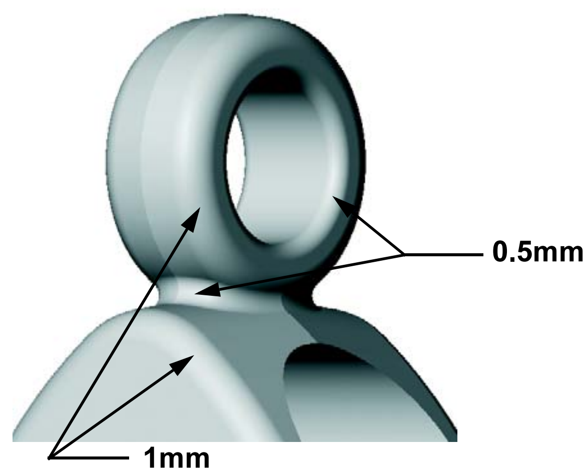
Add a **5mm** diameter hole.

Add the fillet (**1mm**) to the bottom edges after the cut.

Create a third **Through All** cut, **3mm**, centered above the origin.



4 Fillets.
Add fillets of **0.5mm** and **1mm** as shown. Note that these fillets are very order dependent; the **1mm** fillets must precede the **0.5mm** ones.



5 Save and close the part.

Lesson 5
Patterning

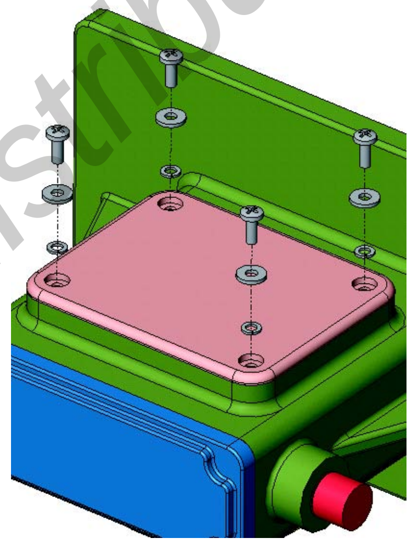
Upon successful completion of this lesson, you will be able to:

- Use several different types of patterns.
- Use geometry patterns properly.
- Use the Vary Sketch option.

Why Use Patterns?

Patterns are the best method when creating multiple instances of one or more features. Use of patterns is preferable to other methods for several reasons.

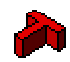
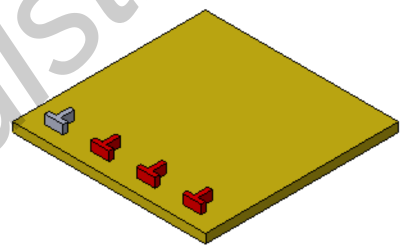
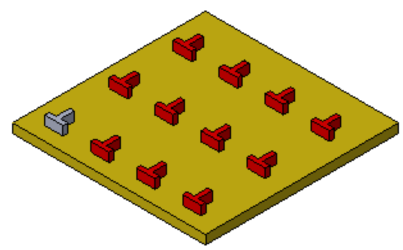
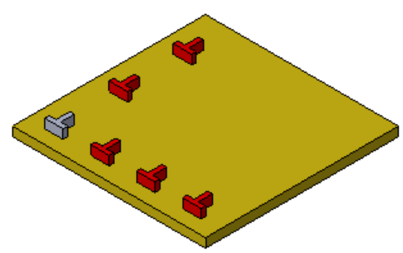
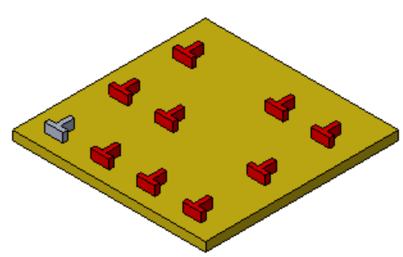
- **Reuse of geometry**
  The original or **seed** feature is created only once. **Instances** of the seed are created and placed, with references back to the seed.
- **Changes**
  Due to the seed/instance relationship, changes to the seed are automatically passed on to the instances.
- **Use of Assembly Component Patterns**
  Patterns created at the part level are reusable at the assembly level as **Feature Driven Patterns**. The pattern can be used to place component parts or sub-assemblies.
- **Smart Fasteners**
  One last advantage with Smart Fasteners to automatically add fasteners to the assembly. These are specific to holes.

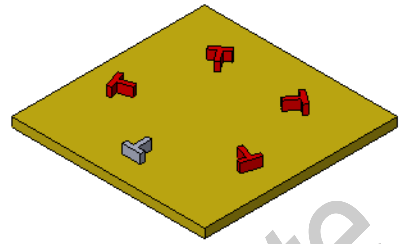
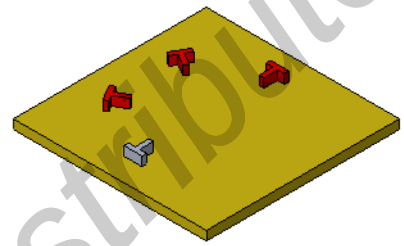
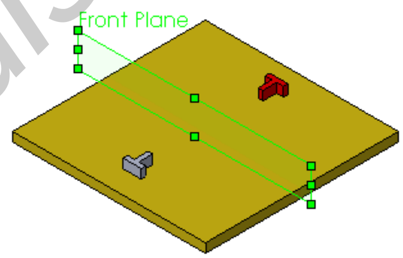
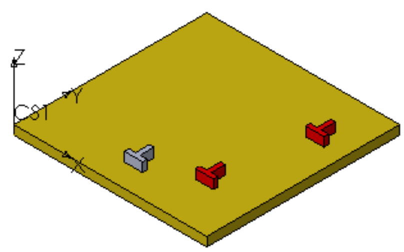
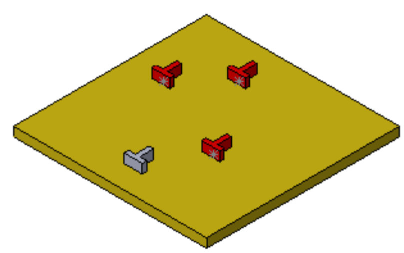
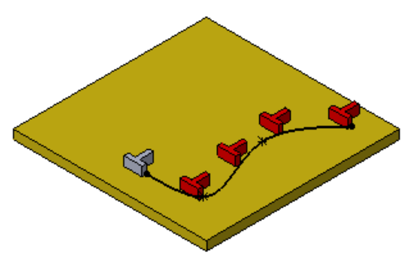


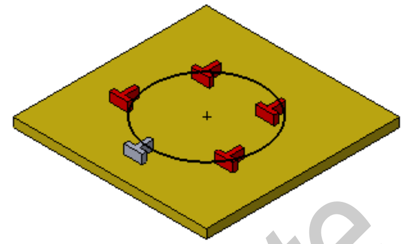
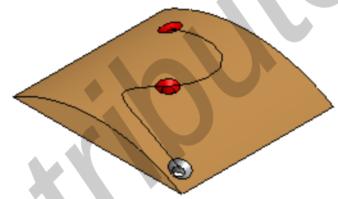
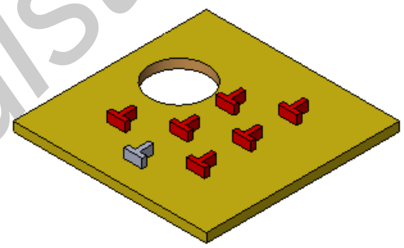
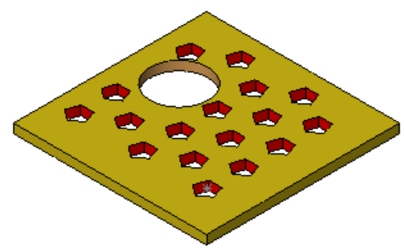
Comparison of Patterns

There are many types of patterns available in SolidWorks and this table is intended to highlight the typical uses for each type.

- **Seed**
  The **Seed** is the geometry to be patterned. It can be one or more features, bodies or faces.
- **Pattern Instance**
  The **Pattern Instance** (or just **Instance)** is the "copy" of the seed created by the pattern. It is in fact much more than a copy because it is derived from the seed and changes with the seed.

Pattern Type:	**Typical usage:**	**Key:** **Seed** =  **Pattern Instance** = 
Linear	One-directional array with equal spacing.	
Linear	Two-directional array with equal spacing.	
Linear	Two-directional array; pattern seed only.	
Linear	One- or two-directional array. Selected instances removed.	

Circular	Circular array with equal spacing about a center.	
Circular	Circular array with even spacing about a center. Selected instances removed or angle less than 360°.	
Mirror.	Mirrored orientation about a selected plane. Can use selected features or the entire body.	
Table Driven . . .	Arrangement based on a table of XY locations from a coordinate system.	
Sketch Driven . .	Arrangement based on the positions of points in a sketch.	
Curve Driven. . .	Arrangement based on the geometry of a curve.	

Curve Driven. . .	Arrangement of full or partial circular path.	
Curve Driven. . .	Arrangement based on the geometry of a projected curve.	
Fill	Arrangement of instances to pattern based on a face.	
Fill	Arrangement of shapes to pattern based on a face.	

Pattern Options

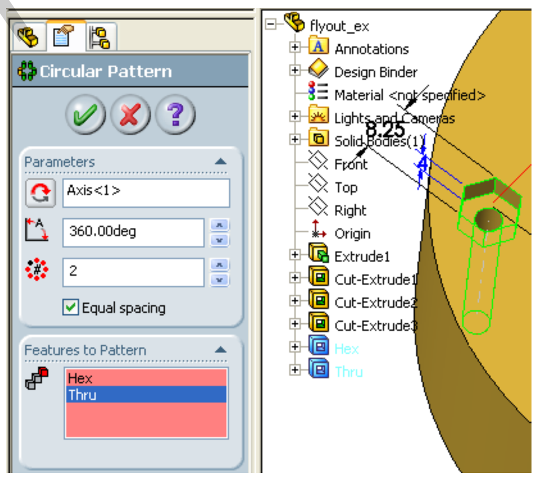
Pattern features share several options. They are unique to this class of feature and will be discussed in detail later in this lesson.

Pattern Feature	Select Feature, Bodies or Faces	Propagate Visual Properties	Pattern Seed Only	Skip Instances	Geometry Pattern	Vary Sketch	Use Shapes
Linear	✓	✓	✓	✓	✓	✓	
Circular	✓	✓		✓	✓		
Mirror	✓	✓			✓		

Table Driven	✓	✓					
Sketch Driven	✓	✓			✓		
Curve Driven	✓	✓	✓	✓	✓	✓	
Fill	Features and Faces only	✓		✓	✓	✓	✓

Flyout FeatureManager Design Tree

The **Flyout** FeatureManager design tree allows you to view both the Feature-Manager design tree and the PropertyManager at the same time. This allows you to select features from the FeatureManager when it would otherwise be obscured by the PropertyManager. It is also transparent, overlaying the part graphics.



The flyout FeatureManager design tree is activated automatically with the PropertyManager. It may appear collapsed and can be expanded by clicking on the plus “+” sign prefix.

Note

The options **Linear Sketch Step and Repeat** and **Circular Sketch Step and Repeat** can be used within a sketch to create copies of sketch geometry. They *do not* create pattern features.

Linear Pattern

The **Linear Pattern** creates copies, or instances, in a linear pattern controlled by a direction, a distance and the number of copies. The instances are dependent on the originals. Changes to the originals are passed on to the instanced features.

Introducing: Linear Pattern

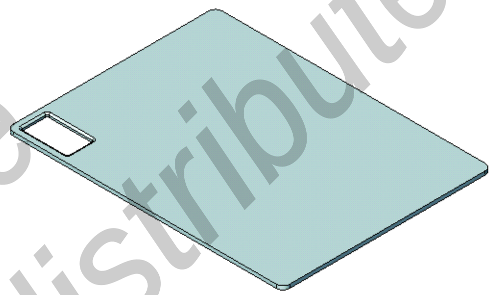
Linear Pattern creates multiple instances in one- or two-dimensional arrays. The axis can be an edge, axis, temporary axis or linear dimension.

Where to Find It

- On the Features toolbar click the **Linear Pattern** tool.
- From the **Insert** menu choose: **Pattern/Mirror, Linear Pattern...**.

1 Open the part named `Grate`.

The part contains the seed feature that will be used in the pattern.



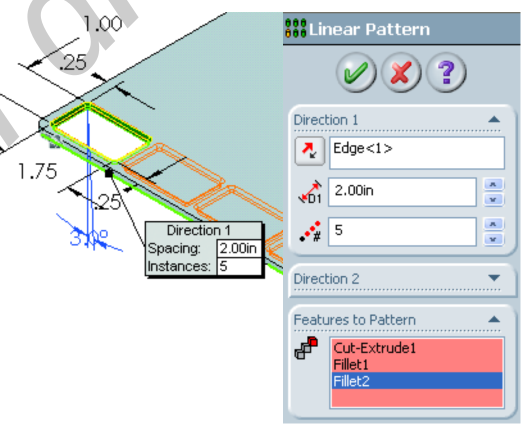
2 Direction 1.

Click **Insert, Pattern/ Mirror, Linear Pattern**.

Select the linear edge of the part and click the **Reverse Direction**, if necessary, to set the direction shown.

Select the three features shown in **Features to Pattern**.

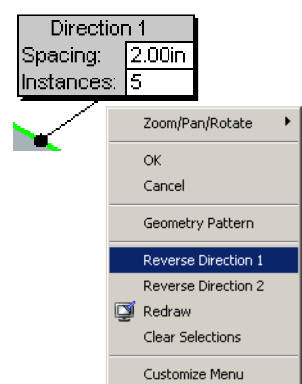
Set the **Spacing** to **2”** and **Instances** to **5**.



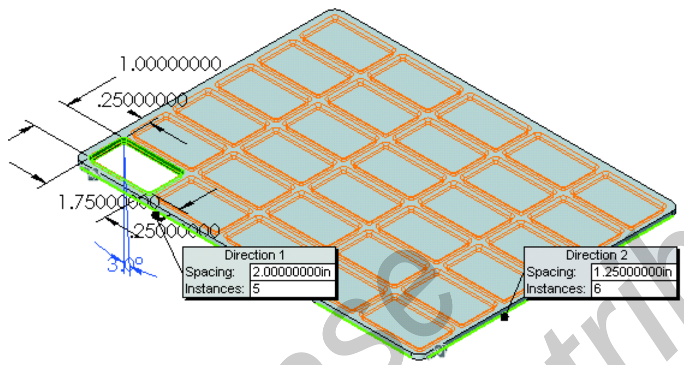
Note

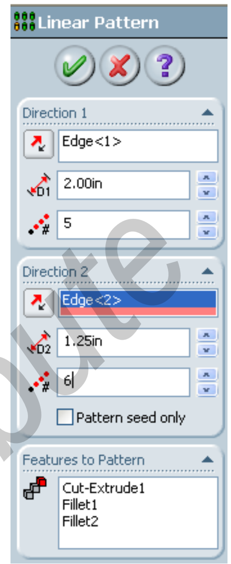
The pattern label is attached to the geometry used to define the pattern direction or axis. It contains the key settings for **Spacing** and **Instances** and is editable. Double-click the setting to change and retype the value.

Right-click the label to access other pattern commands such as **Reverse Direction** and **Geometry Pattern**.



3 Direction 2.
Expand the **Direction 2** group box and select another linear edge.



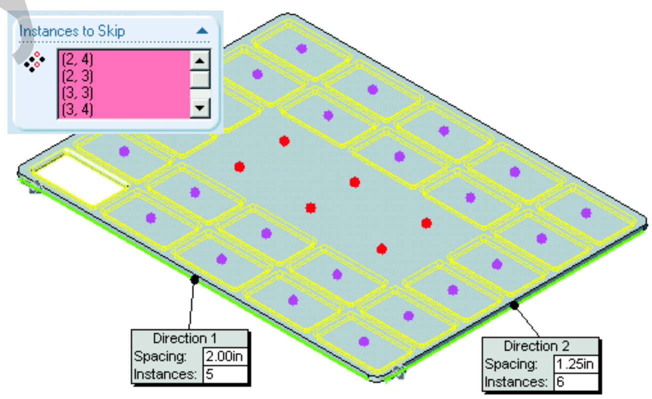


Deleting Instances

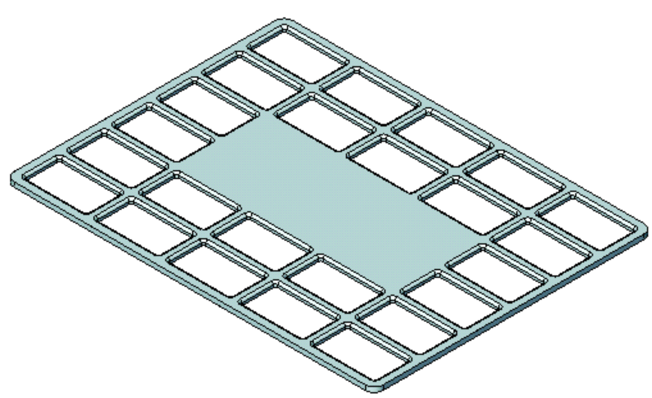
Instances that are generated by the pattern can be deleted by selecting a marker at the centroid of the instance shown in the pattern preview. Each instance is listed in array format **(2,3)** for identification.

The seed feature cannot be deleted.

4 Instances to Skip.
Expand the **Instances to Skip** group box and select the six center instance markers. The tooltip shows an array location that is added to the list when selected.



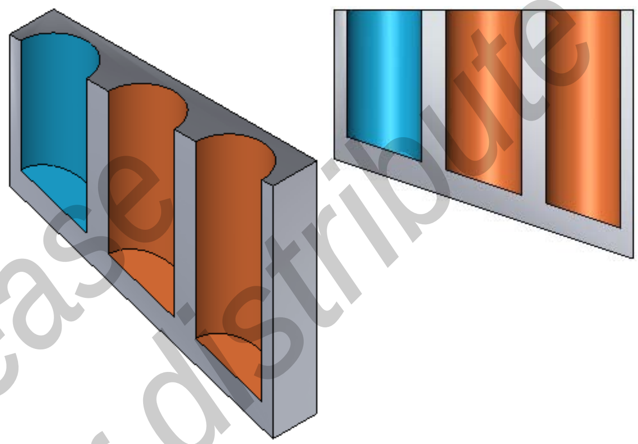
5 Completed pattern.
Click **OK** to add the pattern feature `LPattern1`.



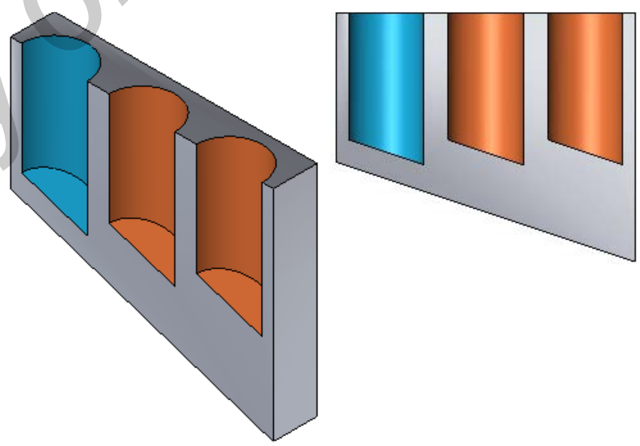
Geometry Patterns

The **Geometry Pattern** option is used to minimize rebuild time by using the **Seed** geometry for all **Instances** in the pattern. It should only be used when the geometry of the seed and the instances are of identical or similar shape.

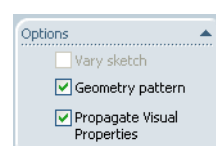
- **Without Geometry Pattern**
  If the **Geometry Pattern** option is *cleared*, the end condition of the seed is used in the instances. In this example, the **Offset From Surface** end condition of the blue seed feature is applied in the orange instances, forcing them to use the same end condition.



- **With Geometry Pattern**
  If the **Geometry Pattern** option is *checked*, the geometry of the seed is used. The geometry is copied along the pattern, ignoring the end condition.



6 Geometry Pattern.
Right-click the Linear Pattern feature and choose **Edit Feature**. Check the **Geometry pattern** option. Because the plate is constant thickness, the resulting geometry will look the same.



Circular Patterns

The **Circular Pattern** creates copies, or instances, in a circular pattern controlled by a center of rotation, an angle and the number of copies. The instances are dependent on the originals. Changes to the originals are passed on to the instanced features.

Introducing: Circular Pattern

Circular Pattern creates multiple instances of one or more features spaced around an axis. The axis can be an edge, axis, temporary axis or angular dimension.

Where to Find It

- On the Features toolbar click the **Circular Pattern** tool.
- From the **Insert** menu choose: **Pattern/Mirror, Circular Pattern...**.

A Word About Axes

Axes are types of **Reference Geometry** that can be used with many pattern features to define direction or rotational axes. There are two types: **Temporary Axes** and **Axes**.

Temporary Axes

Every cylindrical and conical feature has an axis associated with it. View the temporary axes of the part using **View, Temporary Axes**. One axis is displayed through each circular face in the model.

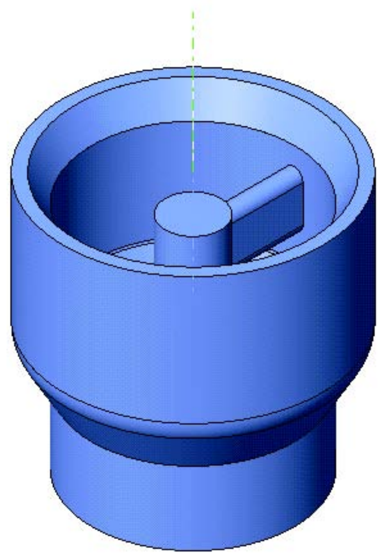
Axes

Axes are features that must be created using one of several methods. The advantages to creating an axis is that it can be renamed, selected by name from the FeatureManager and sized. **Temporary Axes** can be made permanent and given unique names using the **One Line/Edge/Axis** option. See the *Advanced Part Modeling* manual for more information on creating axes.

---

1 **Open the part named `Circular_Pattern`.**

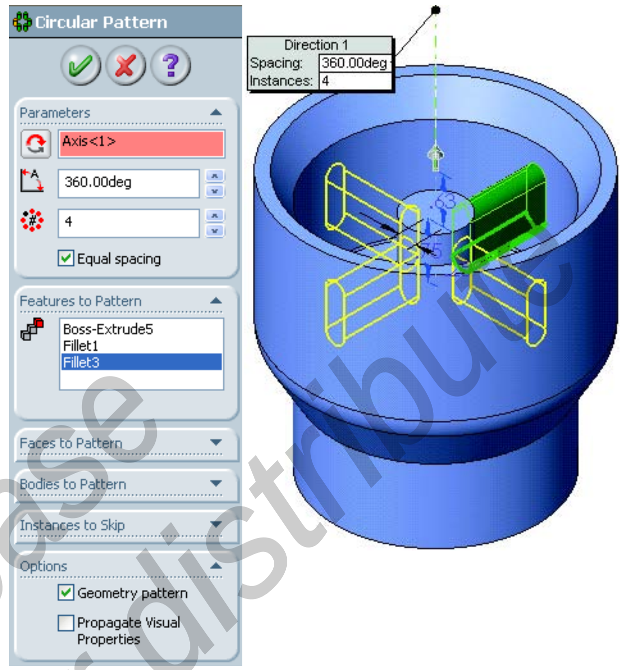
2 **Temporary axes.**
Click **View, Temporary Axes** to see the axes generated automatically for circular features. Select one of the temporary axes.



3 Settings.
Click **Insert, Pattern/Mirror, Circular Pattern...**.

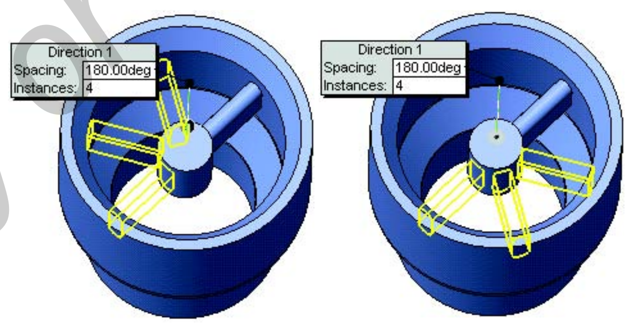
Select the three features shown for **Features to Pattern**. Set the **Angle** to **360°** and **4** instances.

Click **Equal Spacing** and click **Geometry pattern**.



Note

The **Reverse Direction** option is meaningful only when the an angle other than 360° is used.



Mirror Patterns

The **Mirror Pattern** creates a copy, or instance, across a plane or planar face. The instance is dependent on the original. Changes to the original is passed on to the instanced feature.

Introducing: Mirror Pattern

Mirror Pattern creates *one* instance of one or more features or a body across a plane. The plane can be a reference plane or planar face.

Where to Find It

- On the Features toolbar click the **Mirror Pattern** tool.
- From the **Insert** menu choose: **Pattern/Mirror, Mirror...**.

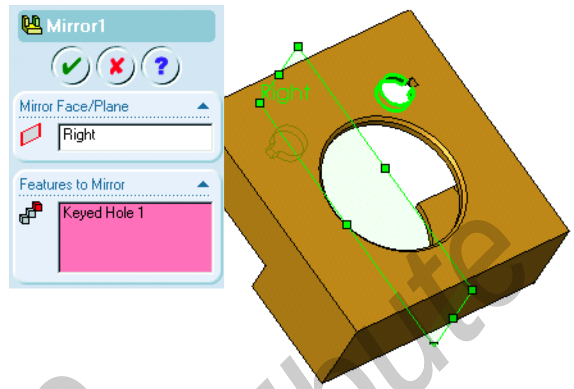
Note

To mirror all the geometry of a part about a common face, select the common face as **Mirror Face/Plane** and the solid body as **Bodies to Mirror**. The common face must be planar.

1 Open the part named `Mirror_Pattern`.

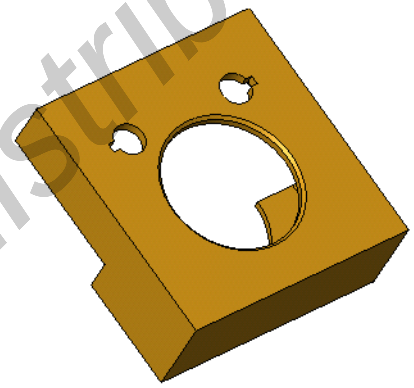
2 Mirror.
Click **Insert, Pattern/ Mirror, Mirror** and the Right plane. Select the library feature as the **Features to Mirror**.

Click **OK**.



Note

Geometry Pattern can also be used with this feature.



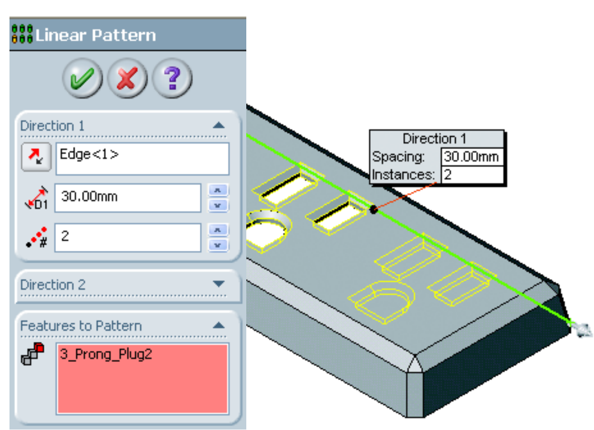
Using Pattern Seed Only

The **Pattern Seed Only** option is used when a two direction pattern is created. The second direction defaults to patterning all geometry created by the first direction unless **Pattern Seed Only** is used to pattern only the original or seed geometry. It is commonly used to prevent overlapping results when the two directions use the same vector.

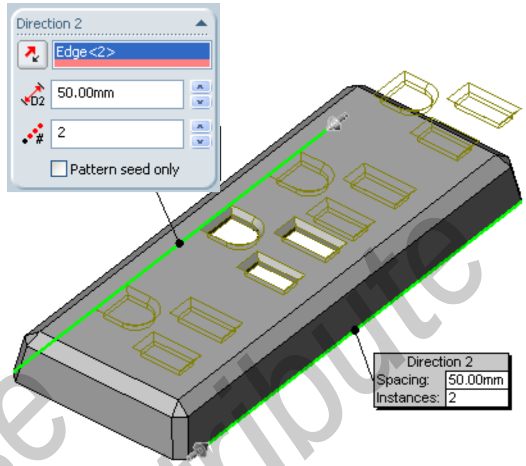
1 Open the part named `Seed_Pattern`.

2 Direction 1.
Click **Insert, Pattern/Mirror, Linear Pattern....** Select the linear edge as the **Pattern direction**, **30mm** as the **Spacing**, **2** as the **Number of Instances**.

For **Features to Pattern**, select the library feature.



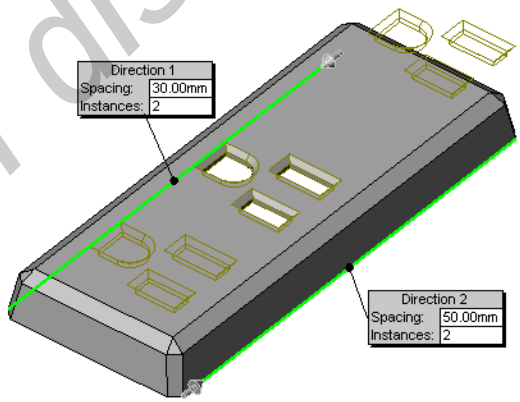
3 Direction 2.
For **Direction 2**, select the linear edge on the opposite side as the direction, reversing the arrowhead. Set the instances to **2**, spacing to **50mm**.

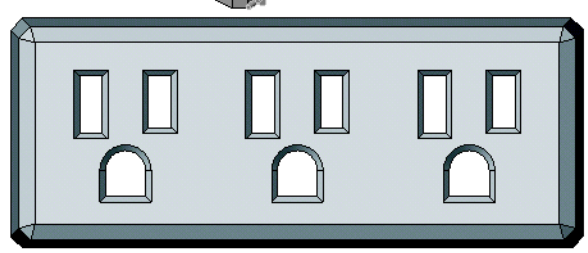


Note As seen in the preview, the original (seed) feature is patterned in both directions.

4 Pattern seed only.
Click **Pattern seed only** to remove the extra instance.

Set the **Direction 2 Spacing** to **30mm**.





Curve Driven Patterns

The **Curve Driven Pattern** creates copies, or instances, in a linear pattern controlled by a curve. The instances are dependent on the originals. Changes to the originals are passed on to the instanced features. The curve can be an entire sketch, edge or single sketch entity. This example will use a sketch that contains several converted model edges.

Tip A 3D curve can be used to drive the pattern. An additional face selection for the **Face normal** is required.

1 Open the part named `Curve_Pattern`.

Introducing: Convert Entities

Convert Entities enables you to copy model edges into your active sketch. These sketch elements are automatically fully defined and constrained with an **On Edge** relation.

Where to Find It

- On the **Tools** menu click **Sketch Tools, Convert Entities**.

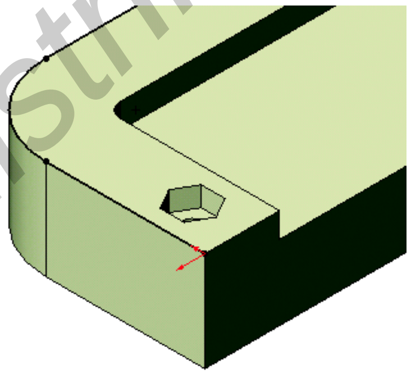
Or, on the Sketch toolbar click **Convert Entities**.

2 Select tangency.

Select the top face and create a new sketch. Right-click the outer edge and select **Select Tangency**.

Click **Convert Entities** to copy the edges into the sketch.

Close the sketch.



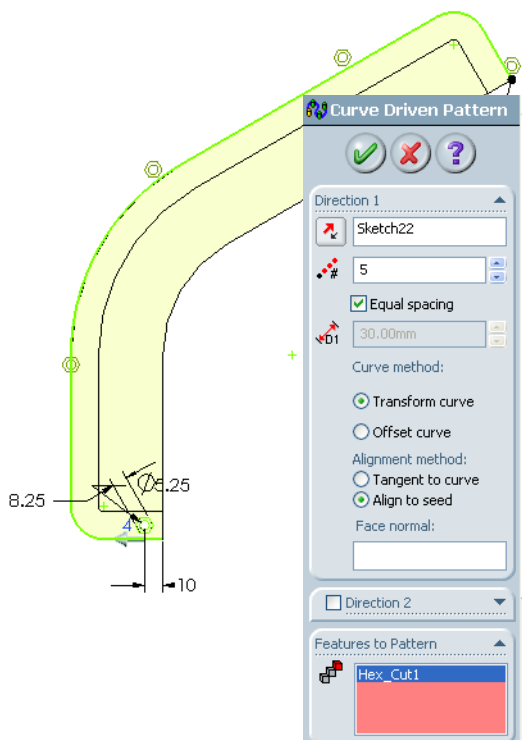
3 Curve driven pattern.

Click **Insert, Pattern/Mirror, Curve Driven Pattern...**.

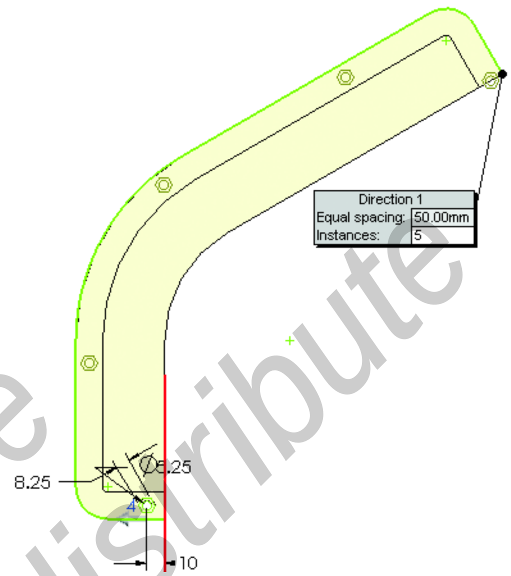
Select the sketch with the converted edges as the **Pattern Direction** and the library feature as the **Features to Pattern**.

Click **Equal Spacing** and set the number of instances to **5**.

Tranform curve and **Align to seed** are used as defaults.

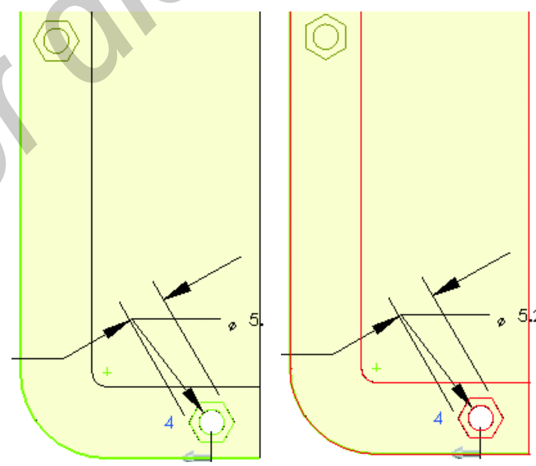


4 Offset curve.
Click the **Curve Method Offset Curve** to position the instances using the same offset as the seed.



5 Align to seed.
Align to seed (leftmost illustration) translates the geometry along the curve direction without rotation of the instances.

Tangent to curve (rightmost illustration) rotates the geometry with changes in the curve direction.



6 Completed pattern.
Click **Tangent to curve** and **OK**. Note that the last instance straddles the edge of the solid body.

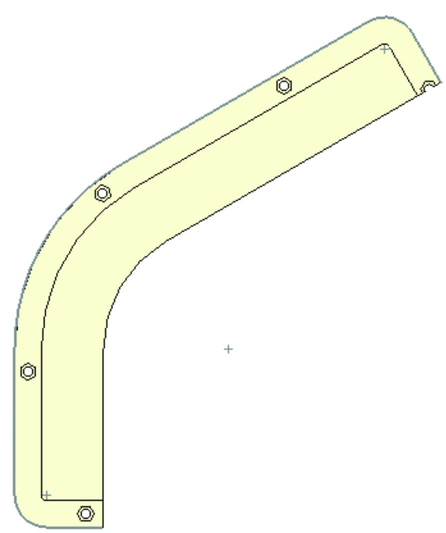
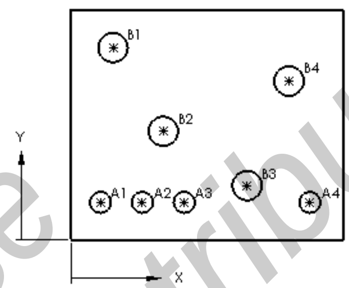


Table and Sketch Driven Patterns

The **Table and Sketch Driven Patterns** creates copies, or instances, in a linear arrangement controlled by sketch points or a table of XY values (Hole Chart). The instances are dependent on the originals. Changes to the originals are passed on to the instanced features.

Tag	XLoc	YLoc	Size
A1	35	45	Ø25 THRU
A2	85	45	
A3	135	45	
A4	285	45	
B1	50	230	Ø35 THRU
B2	110	130	
B3	210	65	
B4	260	190	



Introducing: Sketch Driven Pattern

Sketch Driven Pattern creates multiple instances based on points in a selected sketch. The sketch must exist before the pattern is created.

Where to Find It

- On the Features toolbar click the **Sketch Driven Pattern** tool.
- From the **Insert** menu choose: **Pattern/Mirror, Sketch Driven Pattern...**.

Tip

Only point geometry is used by the Sketch Driven pattern. Other geometry, such as construction lines, can be used but will be ignored by the pattern.

---

1 Open `Table&Sketch_Driven`.

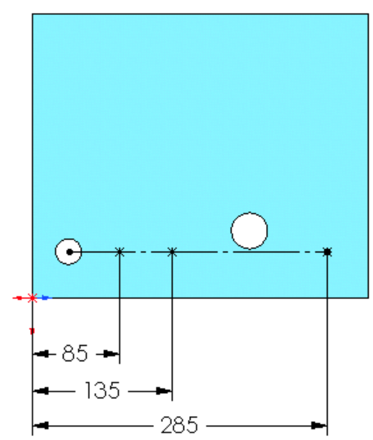
2 Sketch.

Open a new sketch on the face and create a construction line starting at the center of the `D25` feature.

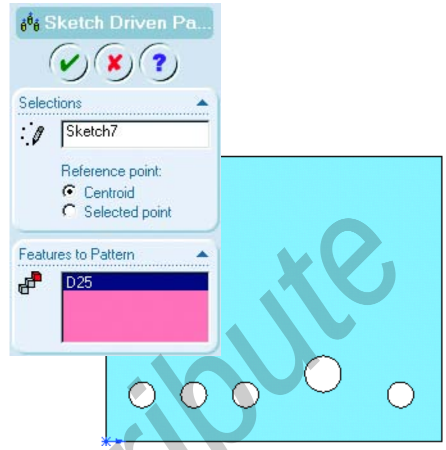
Add points to the sketch and make them coincident with the line.

Dimension and fully define the sketch.

Close the sketch.



3 Sketch driven pattern.
Click **Pattern/Mirror, Sketch Driven Pattern...** and select the new sketch and the **Centroid** option. Under **Features to Pattern**, select the `D25` feature.



Tip The **Centroid** option locates the instances based on the centroid of the seed. If the seed geometry is asymmetric or made up of multiple features, use the **Selected point** option and select a position to use when locating the instances.

Introducing: Table Driven Pattern

Table Driven Pattern creates multiple instances based on a table of XY values. The XY locations are based on a selected **Coordinate System** feature. The Coordinate System must exist before the pattern is created.

Where to Find It

- On the Features toolbar click the **Table Driven Pattern** tool.
- From the **Insert** menu choose: **Pattern/Mirror, Table Driven Pattern....**

Tip The table used in the pattern can be either: typed into the cells in the dialog or be taken from an existing table. The table file type can be either a `*.sldtab` or `*.txt` extension.

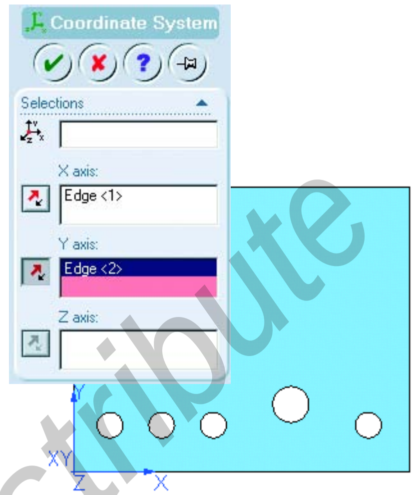
Introducing: Coordinate System

Coordinate Systems create cartesian coordinate systems that can be used for properties, measurements or export output.

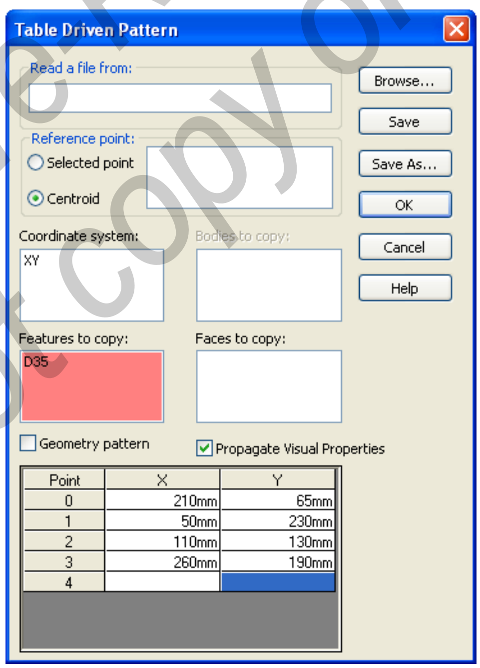
Where to Find It

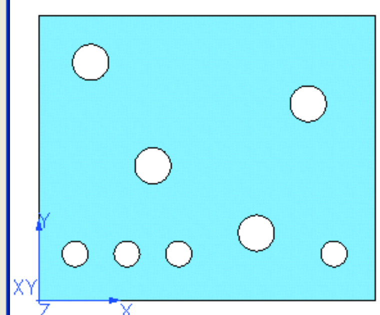
- From the **Insert** menu choose **Reference Geometry, Coordinate System....**
- Or click **Coordinate System** on the Reference Geometry toolbar.

4 **Coordinate system.**
Click **Insert, Reference Geometry, Coordinate System** and select the horizontal edge of the part as the **X axis**. Select the vertical edge as the **Y axis** and use the reverse buttons, if necessary, to reverse the axis directions. Click **OK** and rename the feature `XY`.



5 **Table driven pattern.**
Click **Insert, Pattern/Mirror, Table Driven Pattern...** and select the coordinate system `XY` and `D35` feature. Type in the values for **X** and **Y** in **Point 1**, **2**, **3** and **4** (**0** is the seed).



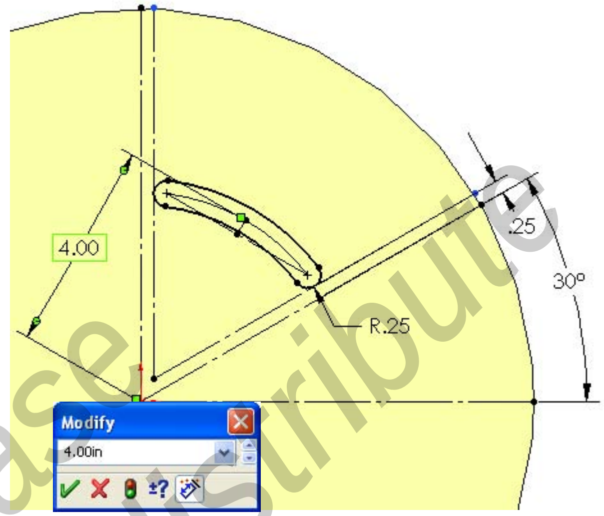


Using Vary Sketch

Vary Sketch is a special case of the **Linear Pattern** that allows instances to change size based on geometric conditions. It requires that the linear dimension driving the shape is selected as the direction of the pattern.

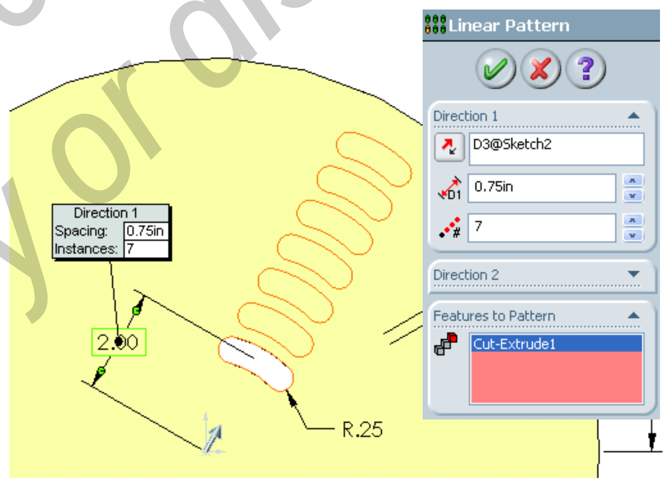
1 Open the part named `Vary_Sketch`.
Edit the sketch for the cut feature. Double-click and change the **2"** dimension, noting that it drives the size of the geometry.

Reset the dimension value.



2 Linear pattern.
Create a linear pattern of the cut feature. Select the **2"** dimension for **Direction 1**. The direction arrow should point away from center.

Enter **0.75"** for **Spacing** and **7** as the **Number of Instances**.

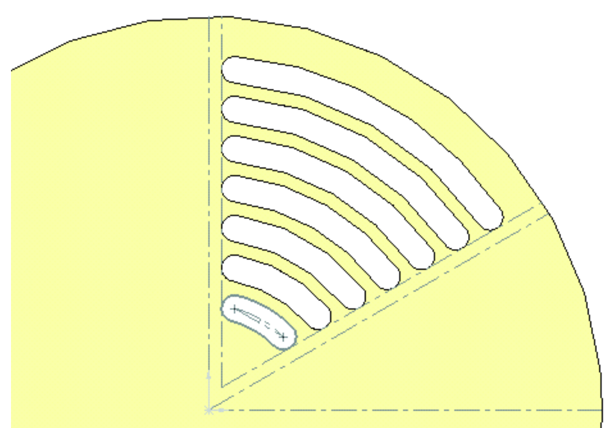


The preview shows that the pattern is created along the measurement direction of the dimension. The size of the cut is unchanged.

3 Select the Vary Sketch option.
The **Vary Sketch** option changes the shape along the direction vector.

Note

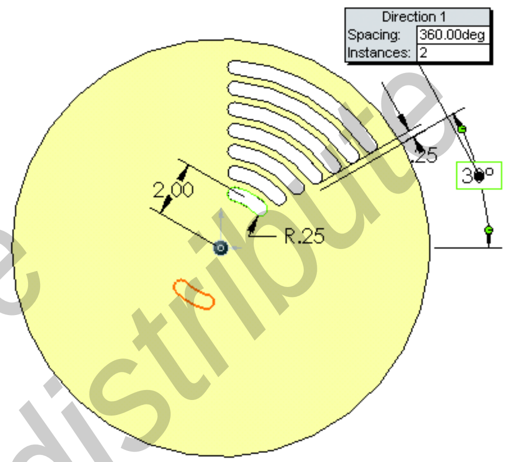
The sketch has been shown for clarity.



Pattern of a Pattern

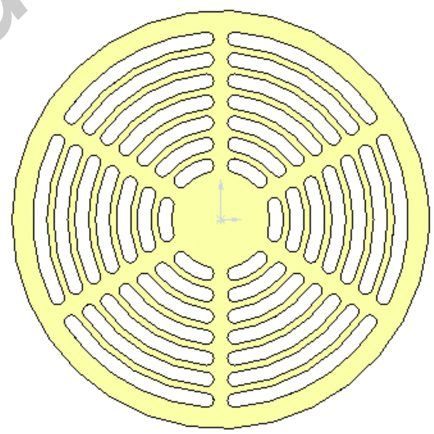
Existing pattern features can be used in new patterns. The original and all pattern instances are used when the pattern feature is selected.

4 Circular pattern.
Double-click the cut feature to expose its dimensions. Insert a **Circular Pattern**. Select the **30°** dimension as the **Pattern Axis**.



5 Pattern feature.
Select the linear pattern feature as the **Features to Pattern**.

Set the **Instances** to **6** with **Equal spacing**.



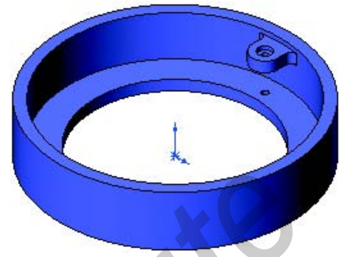
Patterning Faces

Parts that have been imported through IGES, STEP or another method generally appear as a single `Imported` feature. They lack the individual features of native SolidWorks geometry.

Using the **Faces to Pattern** selection, faces can be selected and used as if they were individual features. The selected faces, in conjunction with other part geometry, must form a closed boundary. The result can either add or remove material.

Tip Solid bodies can be patterned using the **Bodies to Pattern** dialog.

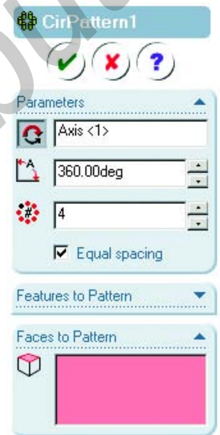
1 Open the existing part `Using Faces in Patterns`.
The part contains imported geometry.



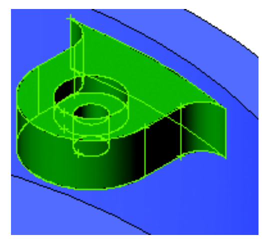
2 Circular pattern.
Click **View, Temporary Axes** and select the center axis as the **Pattern Axis**.

Close the **Features to Pattern** and open **Faces to Pattern**.

Set the pattern to **Equal spacing** with **4** instances.

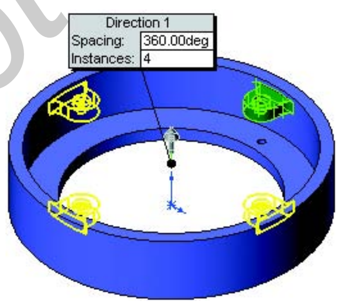


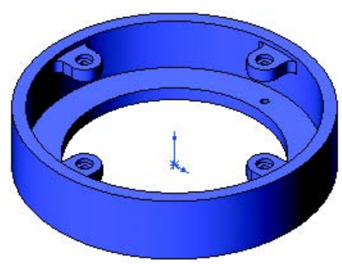
3 Selections.
Select all faces of the object, including the hidden faces.



4 Preview.
The preview displays 4 instances.

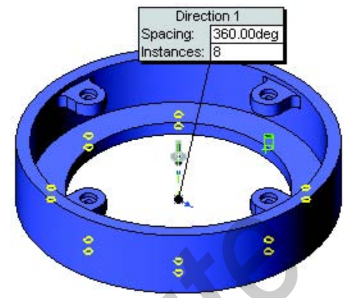
Click **OK** to add the pattern feature.



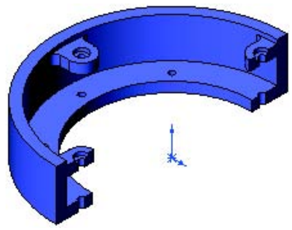


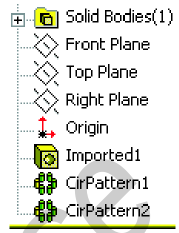
Tip Face selection can be used with any type of pattern. Instances can be skipped.

5 Pattern of "holes".
Create a second pattern using the cylindrical "hole" face of **8** instances.



A section shows how the geometry has been changed by the patterns.



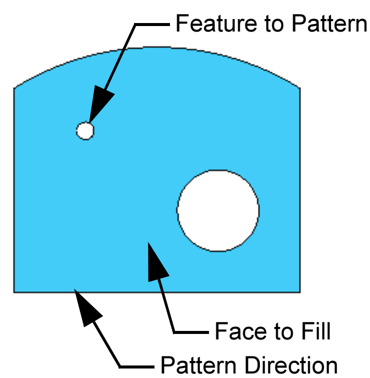


Fill Patterns

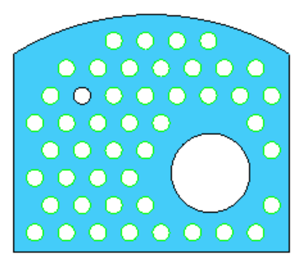
The **Fill Pattern** uses existing features or standard shapes to fill a boundary face. Settings are used to determine the pattern layout, angle, distance and margin distances.

Topological Selections

The topological selections include the faces to fill, the direction and the features to pattern.



- **Face(s) to Fill**
  The face or faces (coplanar) that the pattern will contour to and fill. Loops within the face are not patterned.
- **Pattern Direction**
  A model edge, axis or sketch line that defines the direction of the pattern.
- **Feature(s) to Pattern**
  The selected feature or features to pattern.

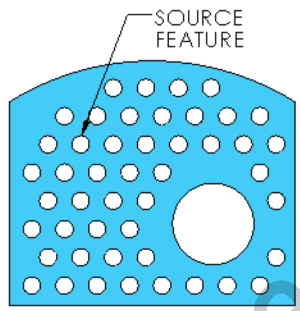
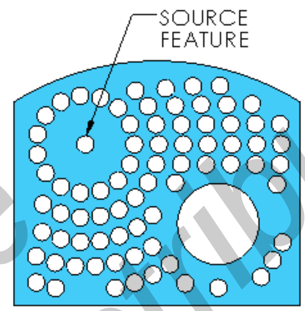
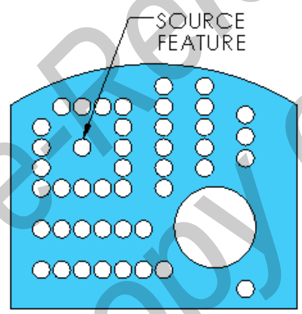
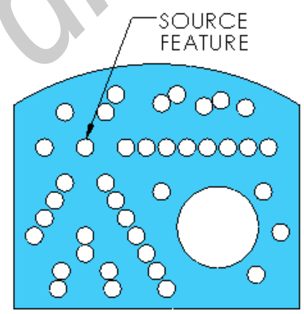


Note

Predefined shapes (circles, squares, diamonds or polygons) can be patterned in place of existing geometry.

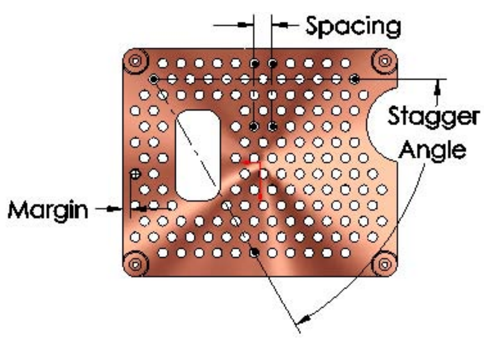
Pattern Layout

The **Pattern Layout** is used to select and arrange the pattern style. With each layout the numeric settings of spacing and distance take on different meanings.

Perforation	**Circular**
	
Square	**Polygon**
	

Numeric Settings

Numeric settings determine the spacings, staggers and minimum edge distances used by the pattern as it fills.



These numeric settings are for the **Perforation** pattern used in the following example.

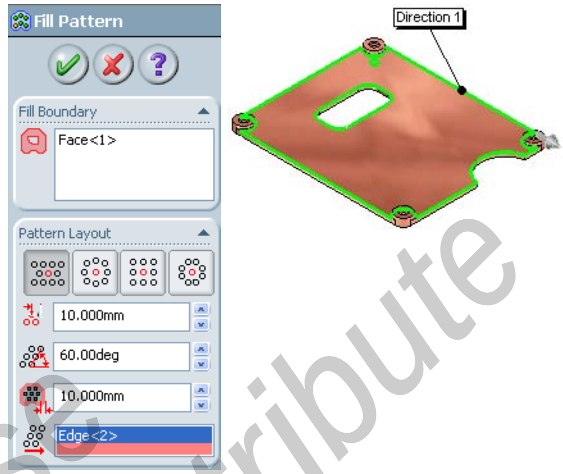
- **Stagger Angle**
  The angle between the selected **Pattern Direction** and the rows of instances.
- **Spacing**
  The minimum distance between similar rows of instances.
- **Margin**
  The minimum distance between the edge of the face and an instance.

1 **Open the existing part `Perforation Area Pattern`.**

2 Face and direction.

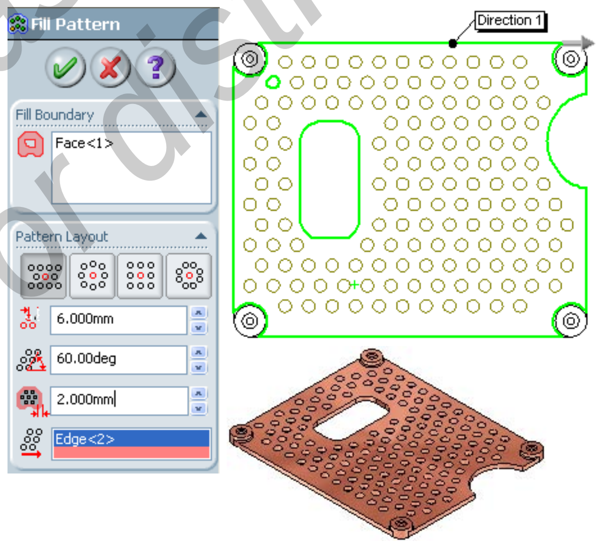
Click **Fill Pattern** and select the top face of the model. Click the edge shown as the direction (**Direction 1**).

Also, select the **Perforation Pattern Layout** button.



3 Settings.

Select the `1/8 (0.125) Diameter Hole1` as the feature to pattern. Set **Spacing**, **Margins** and the **Stagger Angle** using the values shown.



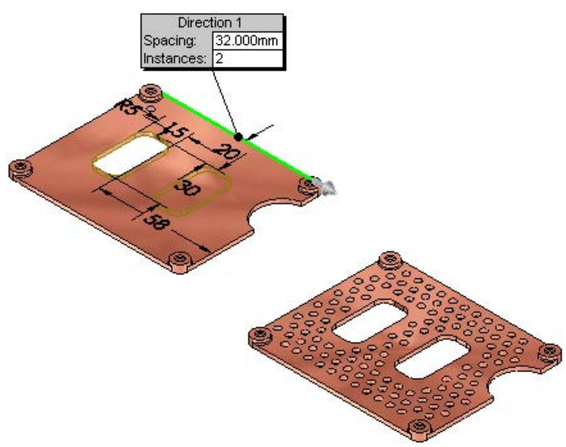
Note

The **Fill Pattern** includes the **Instances to Skip** option like other patterns. It can be used to eliminate any unwanted instances.

4 Changes.

Delete the pattern. Add a **Linear Pattern** to copy the `rectangular cut` as shown.

Recreate the pattern using the same selections. The pattern works around the new hole in the face.



Tip

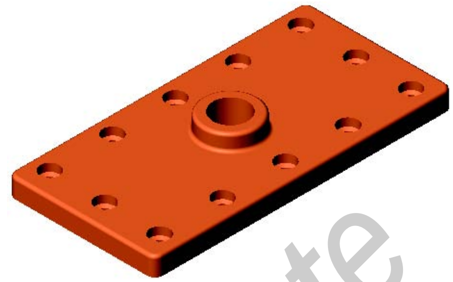
Rather than deleting the feature, **Rollback** could be used to insert the linear pattern prior to the pattern. Rollback will be discussed in *Editing: Repairs*, *Rollback to Feature* on page 241.

Exercise 15: Linear Patterns

Create feature patterns in this part using a Linear Pattern.

This lab uses the following skills:

- Linear patterns.
- Deleting pattern instances.



Procedure

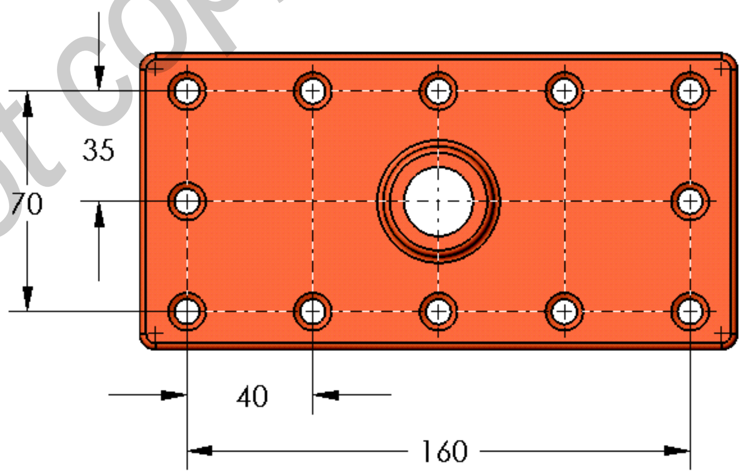
Open an existing part.

Note

This part has been copied for use in linear, table driven and sketch driven patterns.

1 **Open the part `Linear Pattern`.**
The part includes the "seed" feature used in the patterns.

2 **Linear pattern.**
Create a pattern using the seed. Use the dimensions below.



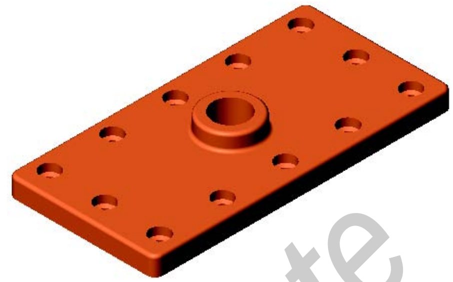
3 **Save and close the part.**

Exercise 16: Table or Sketch Driven Patterns

Create feature patterns in this part using a Table Driven Pattern.

This lab uses the following skills:

- Coordinate systems.
- Table Driven Patterns.
- Sketch Driven Patterns.
- Table files.

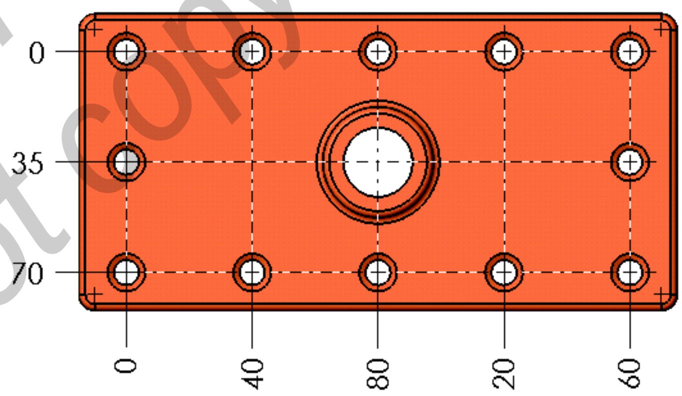


Procedure

Open an existing part.

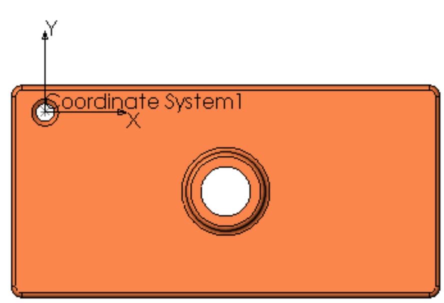
1 **Open the part** **`Table Driven Pattern`.**
The part includes the "seed" feature used in the patterns.

2 **Table or Sketch driven pattern.**
Use the dimensions below to define the sketch used with the Sketch Driven Pattern.



-or-

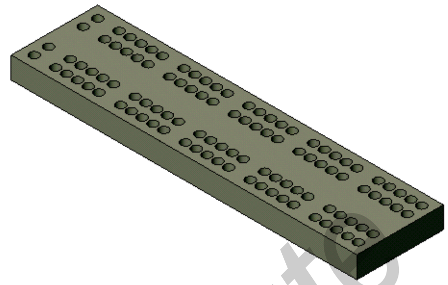
Create a Coordinate System at the center of the `pattern hole` feature and use the files `pattern file.sldptab` or `pattern file.txt` to define the Table Driven Pattern. Create a pattern using the seed feature.



3 **Save and close the part.**

Exercise 17: Skipping Instances

Complete this part using the information and dimensions provided.



This lab reinforces the following skills:

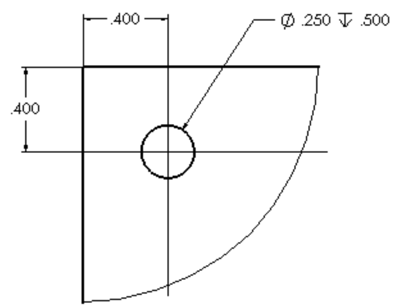
- Creating a linear pattern.
- Skipping instances.
- Patterning a pattern.
- Editing a feature.

Procedure

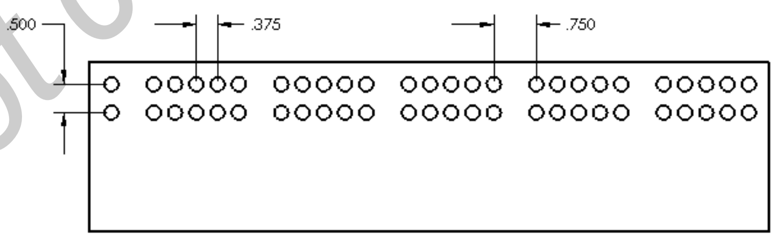
Create a new part with **Inch** units.

1 **Base feature.**
Create a block **3”x12”x0.75”**. It will be useful to have a reference plane centered along the long direction.

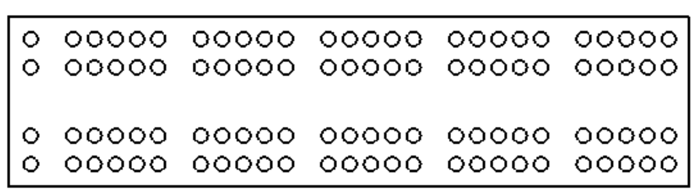
2 **Seed.**
Create the seed feature using the Hole Wizard and an ANSI Inch drill.



3 **Pattern.**
Pattern the hole, skipping instances as shown in the diagram below.



4 **Pattern of a pattern.**
Pattern the pattern to create a symmetrical arrangement of holes.



5 **Change.**
Change the hole to **5/16”** diameter and rebuild.

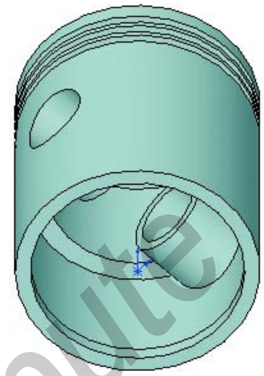
6 **Save and close the part.**

Exercise 18: Linear and Mirror Patterns

Complete this part using the information and dimensions provided.

This lab reinforces the following skills:

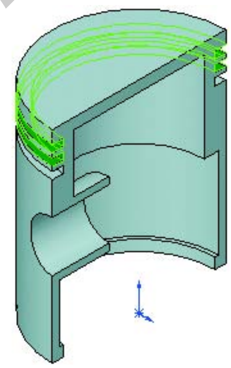
- Creating a Linear Pattern.
- Creating a Mirror Pattern using features.
- Creating a Mirror Pattern using a body.



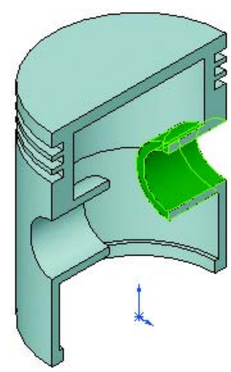
Procedure

Open the existing part `Linear & Mirror`.

1 **Linear pattern.**
Using the existing feature, create a **Linear Pattern** that results in three grooves that are spaced **0.20”**.

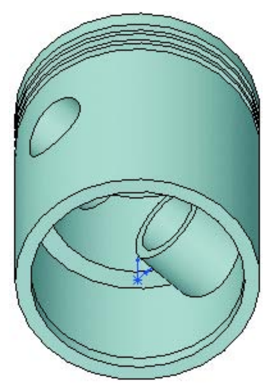


2 **Mirror features.**
Using a single pattern feature, create the duplicate boss and cut as shown.



3 **Symmetry.**
Use a third pattern feature to create the full model from the half model.

4 **Save and close the part.**

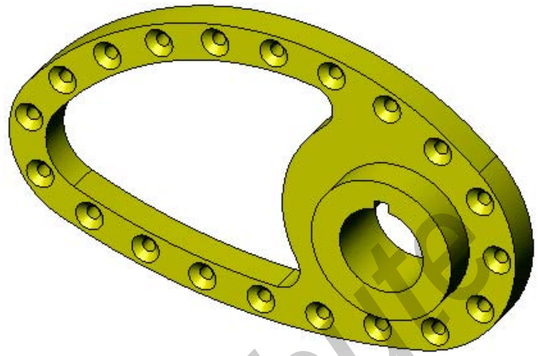


Exercise 19: Curve Driven Patterns

Create feature patterns in this part using a Curve Driven Pattern.

This lab uses the following skills:

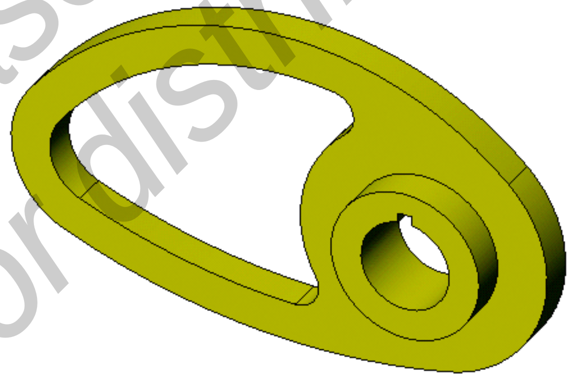
- **Offset Curve**.
- Using the Hole Wizard.
- Curve Driven Patterns.



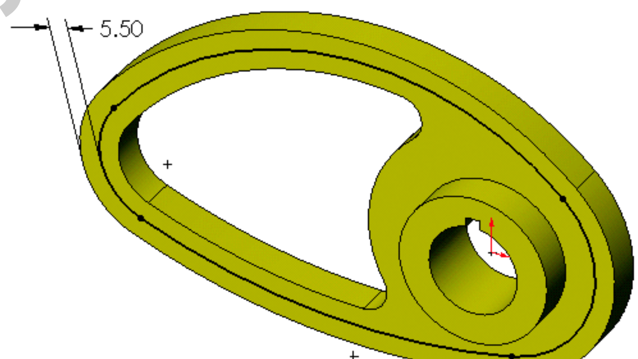
Procedure

Open an existing part.

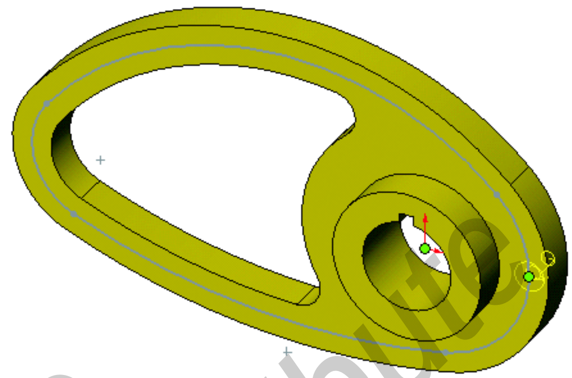
1 **Open the part `Curve Driven Pattern`.**



2 **Sketch.**
Open a sketch on the front face of the cam and create a **5.5mm** offset of the outer profile.
Rename the sketch `Curve Sketch`.



3 Hole Wizard.
Add a Countersink with the following options:

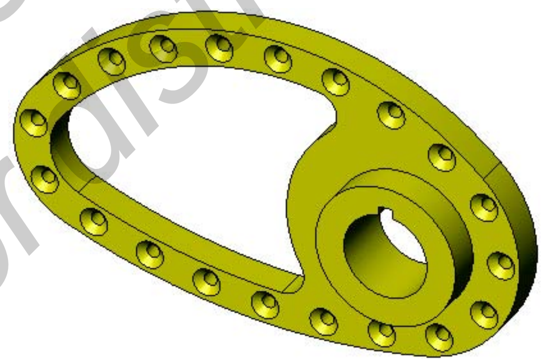


- **Ansi Metric**
- **Flat Head Screw**
- **M3.5**
- **Through All**

Add two geometric relations as follows:

- **Horizontal** with respect to the origin.
- **Coincident** to the `Curve Sketch`.

4 Curve Driven Pattern.
Pattern the feature with the **Number of Instances** to **20** and **Equal spacing**.



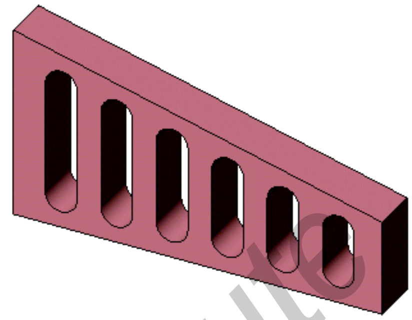
5 Save and close the part.

Exercise 20: Using Vary Sketch

Edit a pattern in this part to use Vary Sketch.

This lab uses the following skills:

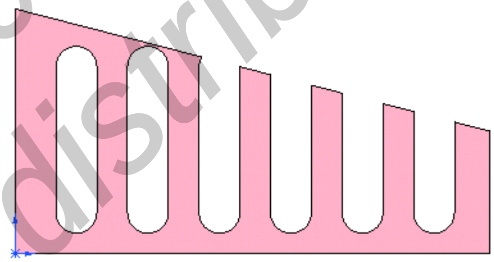
- **Edit Feature**.
- Using the **Vary Sketch** option.



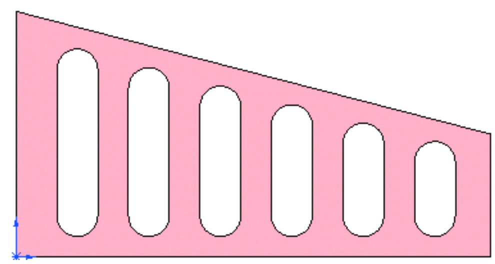
Procedure

Open an existing part.

1 **Open the part `Vary Sketch Lab`.**
The part includes the “seed” feature used in the patterns.

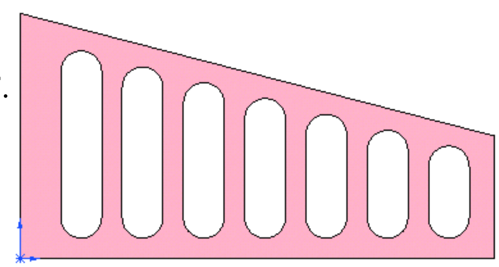


2 **Edit Pattern.**
Edit the pattern feature and make changes to use the Vary Sketch option as shown.



3 **Changes.**
Change the spacing to **0.375”** and the number of instances to **7**.

4 **Save and close the part.**

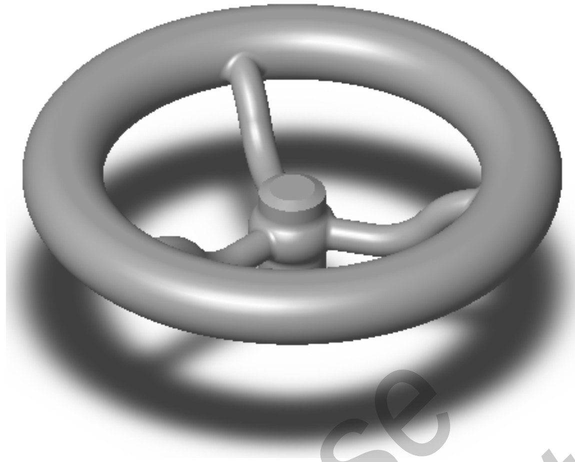


Lesson 6
Revolved Features

Upon successful completion of this lesson, you will be able to:

- Create revolved features.
- Apply special dimensioning techniques to sketches for revolved features.
- Use the multibody solid technique.
- Create a sweep feature.
- Create circular patterns of features.
- Calculate the physical properties of a part.
- Perform rudimentary, first past stress analysis.

Case Study: Handwheel



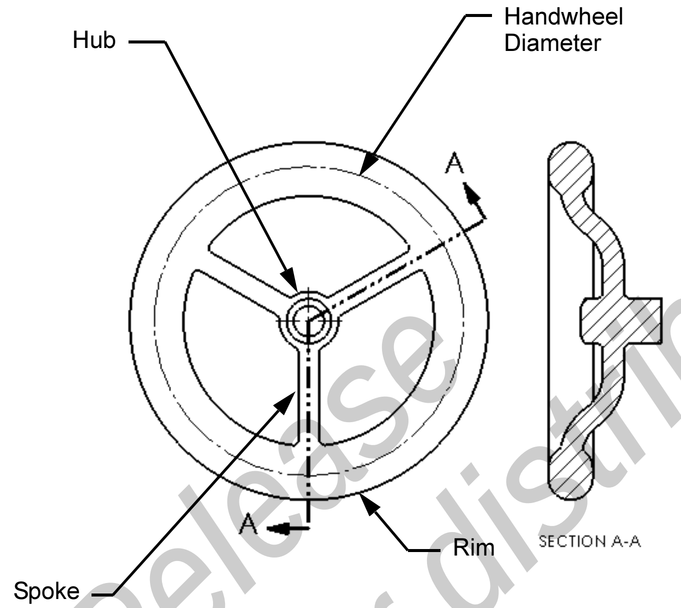
Stages in the Process

Some key stages in the modeling process of this part are shown in the following list.

- **Design intent**
  The part's design intent is outlined and explained.
- **Revolved features**
  The center of the part is the `Hub`, a revolved shape. It will be created from a sketch with a construction line as the axis of revolution.
- **Multibody solids**
  Create two discrete solids, the `Hub` and the `Rim`, connecting and merging them using a third solid, the `Spoke`.
- **Sweep features**
  The Spoke feature is created using a sweep feature, a combination of two sketches that define a sweep profile moving along a sweep path.
- **Circular patterns**
  Rather than model the same spoke multiple times, we will create a pattern of evenly spaced `Spokes` around the centerline of the `Hub`.
- **Analysis**
  Using tools that are included in the SolidWorks software, you can perform basic analysis functions such as mass properties calculations and first-pass stress analysis. Based on the results, you can make changes to the part's design.

Design Intent

The design intent of this part is shown below:



- Spokes must be evenly spaced.
- The center of the rim of the handwheel lies at the end of the spoke.
- The spokes pass through the center of the hub.

Revolved Features

The `Hub` is a revolved feature. It is the first feature created by revolving geometry around an axis. Revolved features require axisymmetric geometry and a centerline (used as the axis) in the sketch. This revolved feature will be used as the center of the wheel. Under the right circumstances, a sketch line may also be used as the centerline.

Procedure

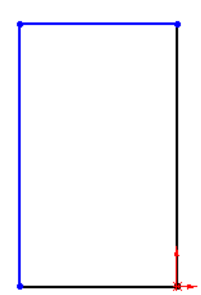
To begin this case study:

1 Open a new part using the `Part_MM` template.

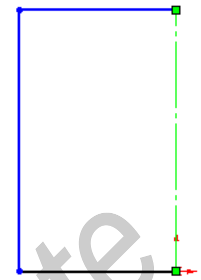
Sketch Geometry of the Revolved Feature

Geometry for the revolved feature is created using the same tools and methods as extruded features. In this case, lines will be used to form the shape – a cylinder with a chamfered edge. The centerline is used as the axis of revolution and for locating geometry.

2 Rectangle.
Right-click the `Right Plane` and select **Insert Sketch**. Create a rectangle from the Origin approximately **50mm** high by **30mm** wide.



3 Convert to construction.
Select the vertical line shown and click **Construction Geometry** on the Sketch toolbar. The line is converted into a construction line.



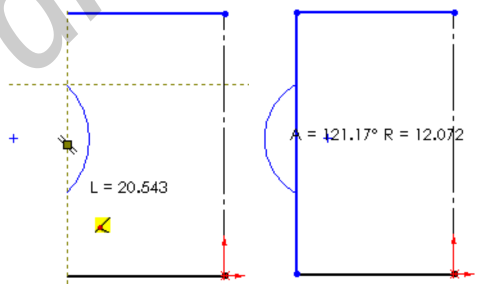
Introducing: 3 Point Arc

The **3 Point Arc** option allows you to create an arc based on three points, the two endpoints followed by a point on the curve.

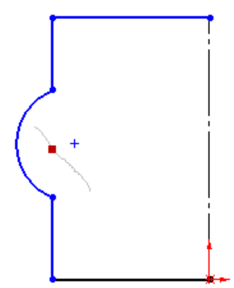
Where to Find It

- From the **Tools** menu choose **Sketch Entities, 3 Point Arc**.
- Or, on the Sketch toolbar click **3 Point Arc**.

4 Insert 3 Point Arc.
Begin the arc by positioning the cursor on the left vertical line and dragging downwards along that edge. Release the mouse button and then select and drag the point on the curve away from the sketch.



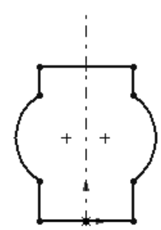
5 Trimming.
Use the **Trim** with the **Power Trim** option and trim away the portion of the line inside the arc.



Rules Governing Sketches of Revolved Features

In addition to the general rules governing sketches that were listed in *Lesson 2: Introduction to Sketching*, some special rules apply to sketches of revolved features:

- A centerline or sketch line must be specified as the axis of revolution.
- The sketch must not cross the axis.


Not Valid

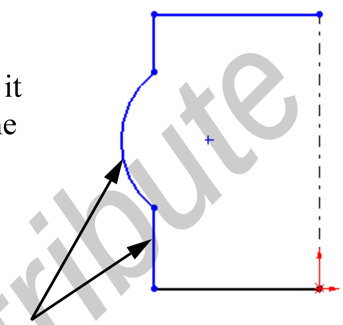
The axis of revolution for the revolve must be selected before creating the revolved feature.

Note that in this example, the right vertical sketch line could be used as the axis of revolution.

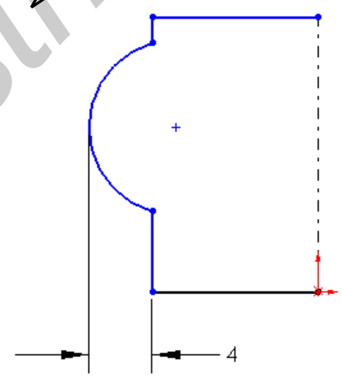
Dimensioning the Sketch

Revolved geometry is dimensioned like any other with one additional option. Dimensions that measure diameters on the finished feature can be changed from linear to diameter dimensions.

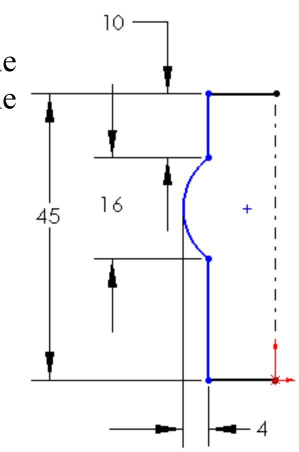
6 Arc dimension.
Dimension the arc by selecting on the circumference of the arc and the vertical line it sits on. The result is a dimension between the line and the tangent of the arc.



7 Finished dimension.
Change the **Value** to **4mm**.



8 Vertical dimensions.
Using the **Vertical Dimension** tool, create the vertical linear dimensions shown at the right. The Smart Dimension icon will also work.



Diameter Dimensions

Some dimensions should be diameter dimensions in the finished revolved feature. For these dimensions, always select the centerline (axis of revolution) as one of the picks. You then have your choice of either a radius or diameter dimension, depending on where you place the dimension text. If you don't pick the centerline, you won't be able to change the dimension to a diameter.

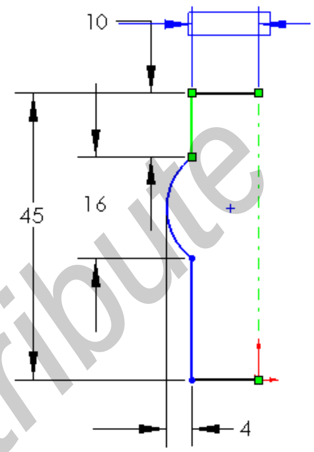
Note

This option is available only if a centerline is used as the axis of revolution. Diameter dimensions are *not* restricted to use in revolved feature sketches.

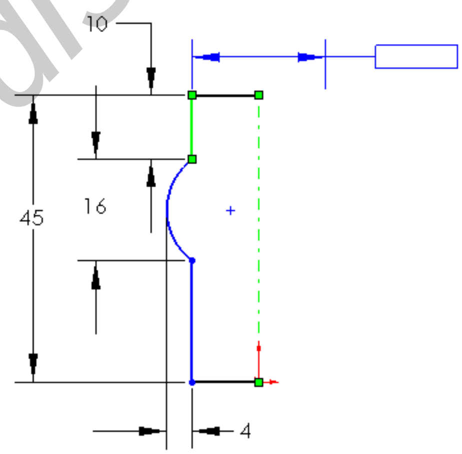
9 Dimension to centerline.
Dimension between the centerline and the outer vertical edge to create a horizontal linear dimension.

Do not click to place the dimension text just yet.

Notice the preview. If you place the text now, you will get a radius dimension.

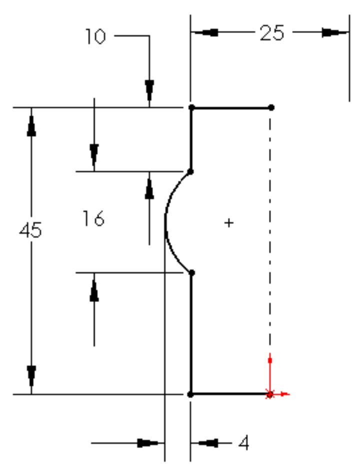


10 Move the cursor.
Move the cursor to the right of the centerline. The preview changes to a diameter dimension.



11 Resulting dimension.
Click to place the dimension text. Change the value to **25mm** and press **Enter**.

Normally, a diameter dimension should have a diameter symbol preceding it thus: ∅25. When the revolved feature is created from the sketch, the system will automatically add the diameter symbol to the **25mm** dimension.



Note

If you inadvertently place the dimension text in the wrong place, and get a radius dimension instead of a diameter you can fix it. Right-click the dimension, and select **Properties...** Click the **Diameter dimension** check box to make the dimension a diameter dimension.

Creating the Revolved Feature

Once the sketch is completed, it can be made into a revolved feature. The process is simple, and a full (360°) revolution is almost automatic.

Introducing: Revolved Feature

The **Revolve** option allows you to create a feature from an axisymmetric sketch and an axis. This feature can be a base, boss or cut feature. The axis can be a centerline, line, linear edge, axis or temporary axis. If only one axis selection is present, it is used automatically. If more than one is present, you must select it.

Where to Find It

- From the **Insert** menu choose **Boss/Base** or **Cut, Revolve...**.
- Or use the Feature toolbar tool: .

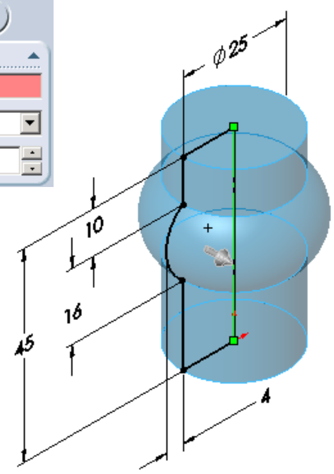
12 Make the feature.
Click **Boss/Base, Revolve...** from the **Insert** menu. A message will appear indicating that the sketch is an open contour and asking if you want to close the contour automatically. Click **Yes**.

The PropertyManager appears with these default end conditions:

One Direction

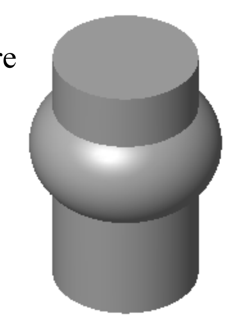
Angle 360°

Accept these defaults by clicking **OK**.



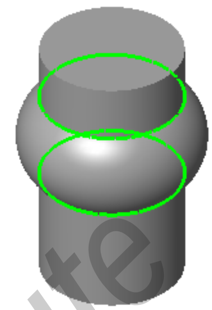
13 Finished feature.
The solid revolved feature is created as the first feature of the part.

Rename it `Hub`.



14 Edit the sketch.

Right-click the `Hub` and select **Edit Sketch**.



Note

You can also right-click the feature in the FeatureManager design tree and achieve the same result.

15 Normal To.

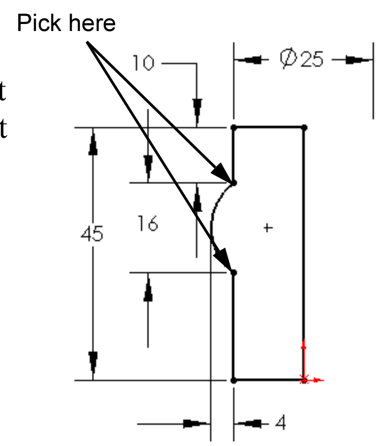
Click **Normal To** on the Standard Views toolbar to change the view so you can see it true size and shape.

16 Fillet settings.

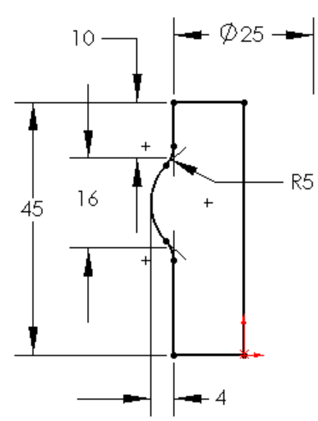
Select the tool and set the value to **5mm**. Make sure the **Keep constrained corners** option is checked.

17 Selections.

Select both endpoints of the arc, as indicated. When each is selected, the fillet will appear. The dimension drives both but only appears once, at the first selection.



Since the endpoints that were filleted had dimensions, **Virtual Sharp** symbols are added where the corners were. These symbols represent the missing corners and can be dimensioned to or used within relations.

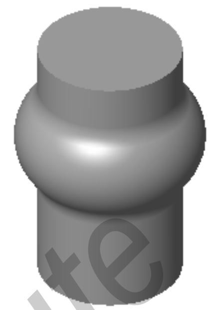


Note

Notice the **25mm** dimension. As mentioned in step **11** on page 183, a diameter symbol now precedes the dimension.

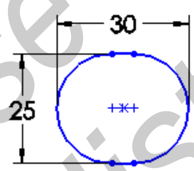
Close the PropertyManager.

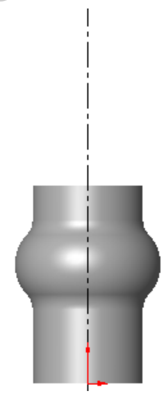
18 Rebuild the model.
To cause the changes to take effect click the **Rebuild** tool.



Building the Rim

The `Rim` of the `Hand-wheel` is another revolved feature. It too is revolved 360°. The profile of the `Rim` is an oval shape, made up of two 180° arcs and two lines.

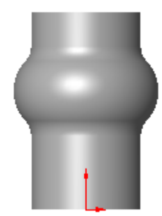




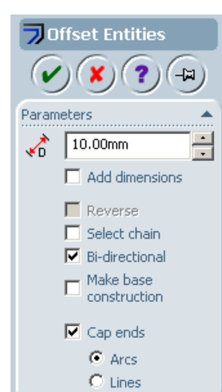
The `Rim` will be created as a separate solid body, not merged to the `Hub`.

19 Sketch.
Create a new sketch on the `Right` reference plane. Orient the model in the same direction.

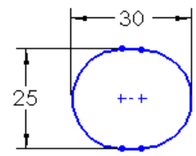
20 Horizontal centerline.
Sketch a short horizontal centerline somewhere off of the model.



21 Offset with cap ends.
The **Offset Entities** tool has a **Cap ends** option that creates **Arcs** to close the ends of a **Bi-directional** offset. Select the centerline and offset using the options shown.



22 Add dimensions.
Dimension the sketch as shown in the illustration at the right.



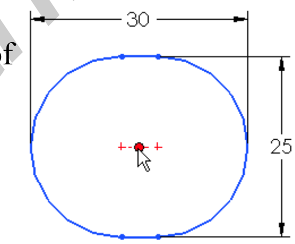
Introducing: Point

The sketch entity **Point** can be used to locate a position in a sketch that other geometry (endpoints for example) cannot.

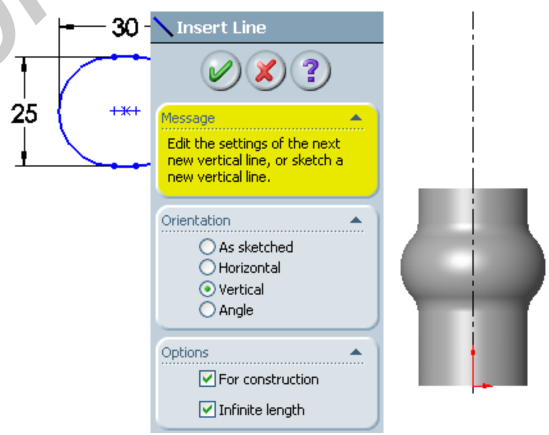
Where to Find It

- Click **Point** on the Sketch toolbar.
- Or from the **Tools** menu, click **Sketch Entities, Point**.

23 Add a point.
Click **Point** and add a point at the midpoint of the centerline.

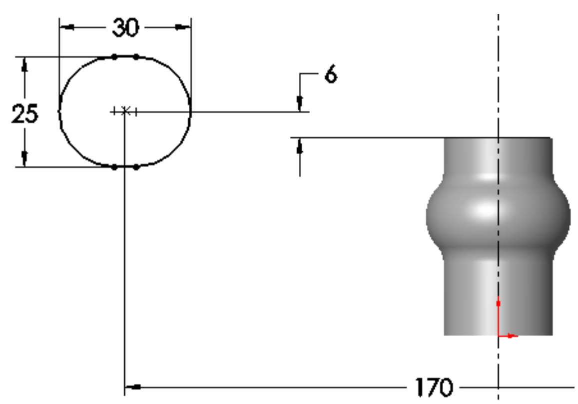


24 Rotation axis.
Add a centerline using the **Centerline** tool, setting **Vertical** and **Infinite length**. Place the line at the origin. This will be the axis of revolution for the revolved feature.



25 Add dimensions.
Add dimensions from the centerline to the point and the arc center to the `Hub` edge.

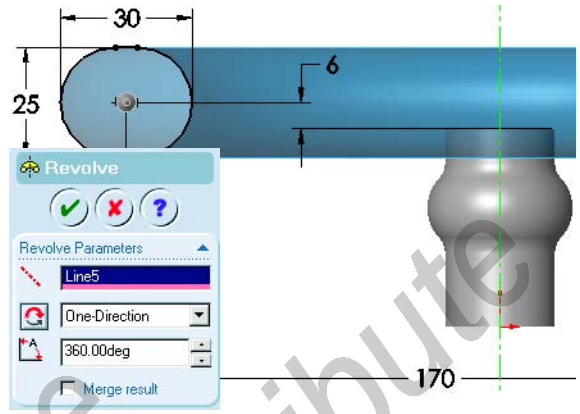
The sketch is now fully defined.



26 Potential ambiguity.
This sketch contains two centerlines. The system will not know which centerline is intended to be the axis of revolution. The centerline to be used can be selected either before or after selecting the **Revolve** tool.

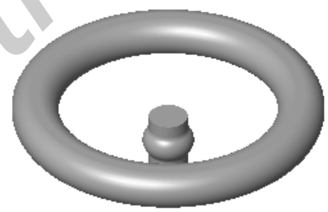
27 Completed feature.
Select the infinite vertical centerline. From the **Insert** menu, choose **Boss/Base, Revolve...**. Use an angle of **360°**.

Rename the feature to `Rim`.



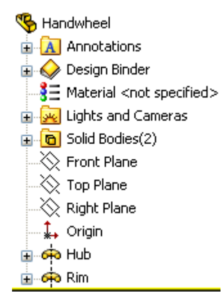
Multibody Solids

Multibody solids occur when there is more than one solid body in a part. In cases where discrete features are separated by a distance, this can be the most efficient method in designing a part.



The `Solid Bodies` folder holds the bodies and also lists how many bodies are currently housed in the folder `(2)`. The bodies can be merged or combined later to create a single solid body.

For more information on multibody parts, see the *Advanced Part Modeling* training manual.



Building the Spoke

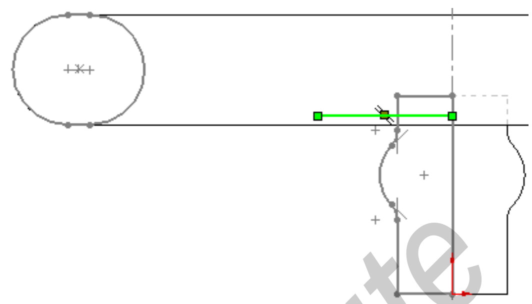
The `Spoke` feature is created using an **Sweep** feature. The sweep pushes a closed contour Profile along an open contour Path. The **Path** is sketched using lines and tangent arcs. The profile is then sketched using a circle. The feature will bridge the space between the existing `Hub` and `Rim` features and combine them into a single solid body.

The `Spoke` feature is important because it will be patterned to create any number of evenly spaced spokes.

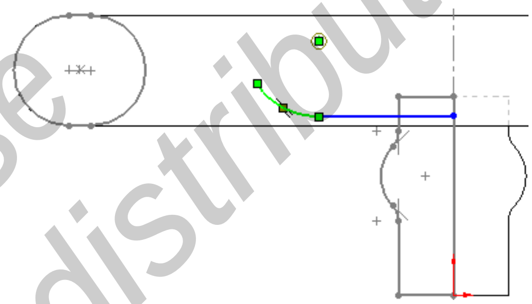
28 Setup.
Setup for sketching:

- Create a new sketch using the `Right` reference plane.
- Show the sketches of the `Hub` and `Rim`.
- Change the display to **Hidden Lines Visible**.

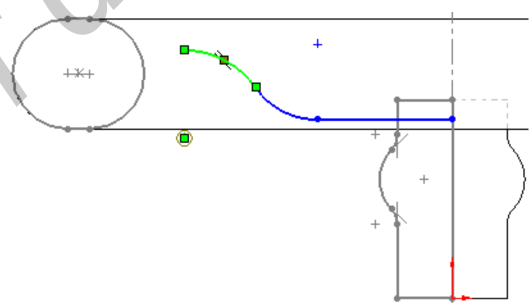
29 Sketch line.
Sketch a horizontal **Line** running from the centerline inside the `Hub` boundaries.



30 Tangent arc.
Create a **Tangent Arc** from the line endpoint in the direction shown. The actual values are not important as you sketch. They will be defined by dimensions later.

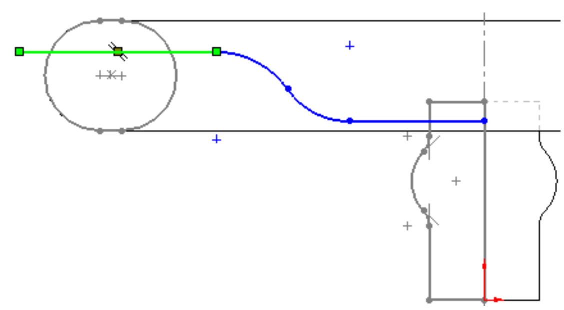


31 Connecting tangent arc.
With **Tangent Arc** still selected, continue sketching by using the previous arc's endpoint as a start. Sketch this arc tangent to the first, ending at a horizontal tangency position.



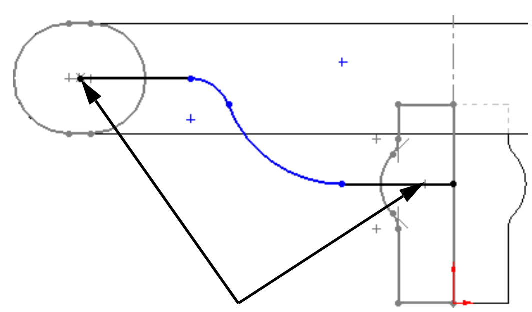
Tip When the vertical inference line coincides with the arc's center, the tangent of the arc is horizontal.

32 Horizontal line.
Sketch a final **Line**. It is horizontal, with its length to be determined by dimensioning.



33 Relations.
Drag and drop the left endpoint of the line onto the point of the `Rim` sketch. A **Coincident** relation is added.

Add another relation between the line at the opposite end and the centerpoint of the arc.

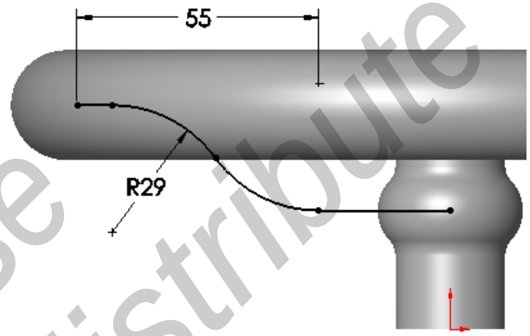


34 Return to a shaded display.

Completing the Path and Profile Sketches

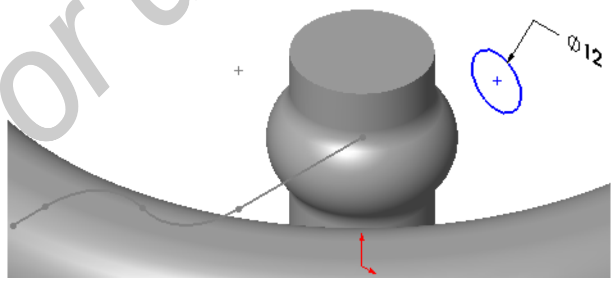
The geometry sketched will act as the "centerline" for the profile sketch.

35 Add dimensions.
Add an **Equal** relation to the arcs. Dimensions are added to define the shape. Picking end points and center points allows for more options when the creating the dimensions.

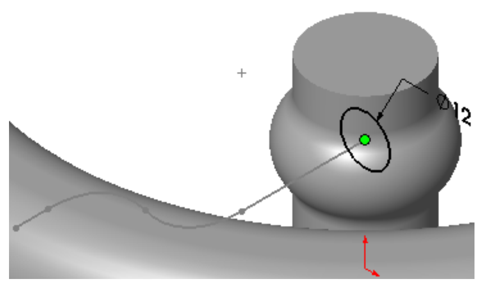


36 Exit sketch.
Right-click in the sketch and choose **Exit Sketch** to close the sketch without using it in a feature.

37 Profile.
Sketch a circle on the `Front` plane and dimension the diameter.



38 Drag relation.
Drag the centerpoint of the circle and drop it on the endpoint of the line in the previous sketch. A coincident relation is added between them. Exit the sketch.



Introducing: Insert, Boss, Sweep

Insert, Boss, Sweep creates a feature from two sketches: a sweep section and sweep path. The section is moved along the path, creating the feature.

Where to Find It

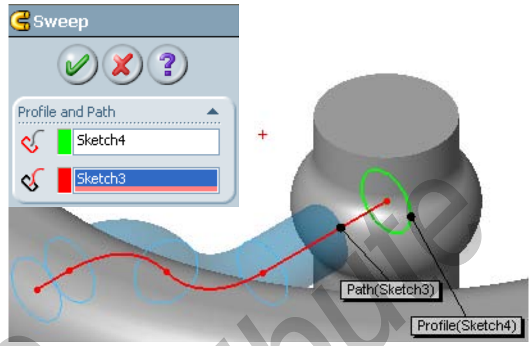
- Click **Sweep Boss/Base** on the Features toolbar.
- Or, click **Insert, Base/Boss, Sweep**.

Note

The **Sweep** command is covered in depth in the *Advanced Part Modeling* course.

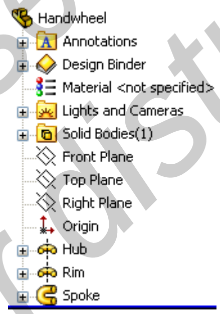
39 Sweep.

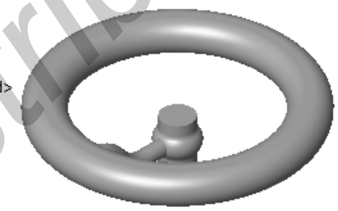
Click the **Sweep** icon and select the closed contour sketch as the **Profile** and the open contour sketch as the **Path**.



40 Results.

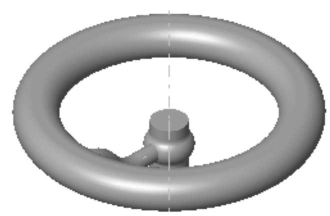
Name the new feature `Spoke`. The `Solid Bodies(1)` folder reflects the merging of solid bodies into one.





41 Temporary axes.

Display the temporary axes using **View, Temporary Axes**.

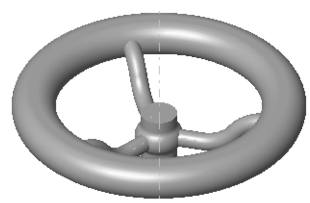


42 Pattern the Spoke.

Click **Circular Pattern** . Select the temporary axis as the center of rotation for the pattern.

Click in the **Features to Pattern** list to make it active. Select the `Spoke`.

Set the **Number of Instances** to **3** with **Equal spacing**.



Rotate View

The **Rotate View** tool allows you to rotate the view of the model freely. To restrict that motion, you can choose an axis, a line or edge, a vertex, or a plane. Click the **Rotate View** tool and the center axis.

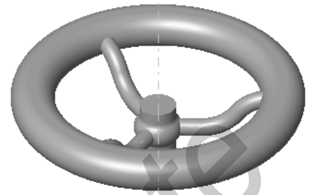
The same result can be obtained using the middle mouse button rotation. Select the temporary axis using the middle mouse button, and drag with the middle mouse button.

Note

If you turned off the temporary axes after you made the circular pattern, you will either have to turn them back on or show the `Rim` sketch in order to have an axis or line (centerline) to rotate about.

43 Rotate.

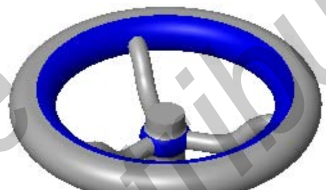
Rotate about the axis by dragging the mouse. Switch axes by simply clicking another axis or other acceptable choice.



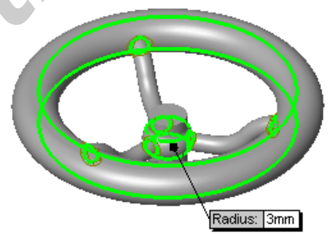
Turn off the Temporary Axes.

44 Add fillets.

To complete the model, **3mm** fillets are added to the highlighted *faces* of the model. Selection of a face selects all edges of that face.



Face selections make the model better suited to withstand dimensional changes.



Chamfers

Chamfers create a bevel on the edge of a model. In many ways, chamfers are similar to fillets in that you select edges and/or faces in the same way.

Introducing: Chamfer

Chamfer creates a bevel feature on one or more edges or vertices. The shape can be defined by two distances or a distance and an angle.

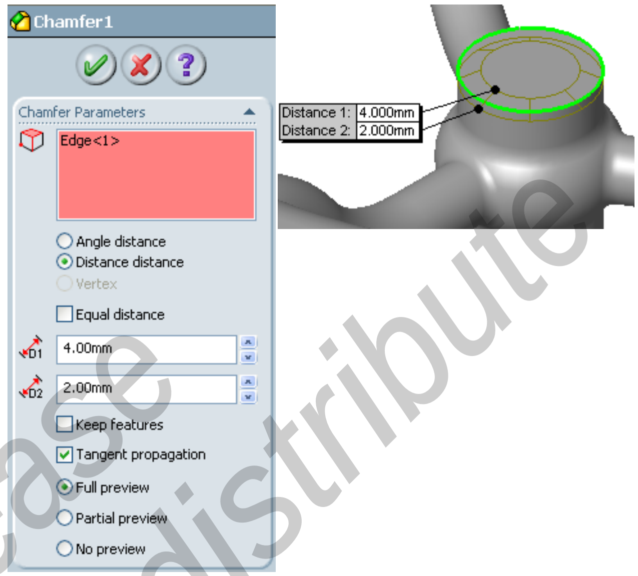
Note

Sketch Chamfers can be added to the sketch rather than to the faces and edges of the solid model.

Where to Find It

- From the **Insert** menu choose **Features, Chamfer...**.
- Or, on the Features toolbar, pick the **Chamfer** tool.

45 Chamfer.
Add a **Chamfer** feature using the top edge of the `Hub` feature. Set the distances using the values shown at right.



Edit Material

The **Edit Material** dialog is used to add and edit the material associated with a part. The material is used for calculations that rely on material properties, including **Mass Properties** and **COSMOSXpress**. The material can vary by configuration. For more information on configurations, see *Lesson 9: Configurations of Parts*.

Where to Find It

- Click **Edit Material** on the Standard toolbar.

RealView Graphics

If you have an NVIDIA graphics accelerator, you may be able to use the **RealView Graphics** option. It provides high-quality, real time material shaders when available.

Tip

Part templates (`*.prtdot`) can include a predefined material.

Where to Find It

- Click **RealView** on the View toolbar.

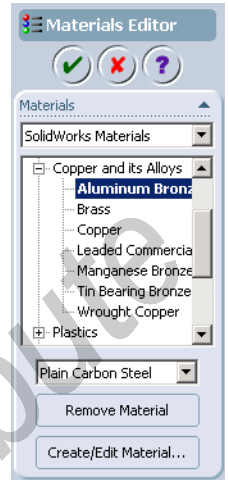
Note

If **RealView Graphics** are not available, the icon will be grayed out.

46 Open `HW_Analysis`.
Close the current part and open the existing part `HW_Analysis`. This part has additional features needed for use in the analysis section of this lesson.

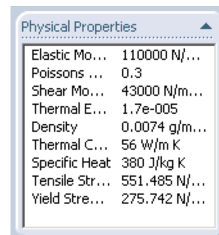
47 Materials.

Click the **Edit Material** icon and select **Copper and its Alloys**, **Aluminum Bronze** from the **Materials** group box.



48 Physical properties.

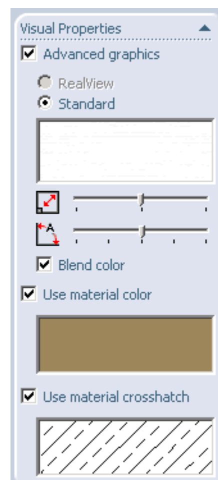
The **Physical Properties** are those assigned by the chosen material.



49 Visual Properties.

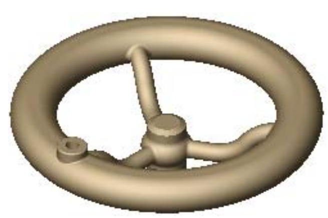
The **Visual Properties** are those assigned by the chosen material. This includes a **Material color**, **Texture** and associated **Material crosshatch**.

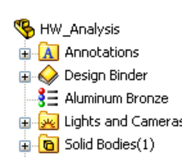
If **RealView** graphics are not available, the option, under **Advanced graphics**, is grayed out and the **Standard** option is chosen.



A change in material changes the color of the part, unless **Use material color** is unchecked.

The material name is also updated in the FeatureManager.





Mass Properties

One of the benefits of working with a solid model is the ease with which you can perform engineering calculations such as computing mass, center of mass, and moments of inertia. The SolidWorks software does all this for you with a simple click of the mouse.

Note

Section Properties can also be generated from a planar face or a sketch in a model. The sketch can be active or selected.

Introducing: Mass Properties

Mass Properties is used to generate the mass properties of the entire solid. The properties include mass, volume and a temporary display of the principal axes.

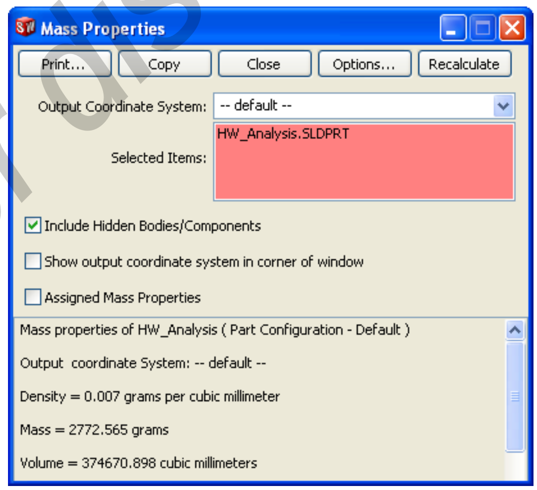
Where to Find It

- On the Tools toolbar click the **Mass Properties** tool.
- From the **Tools** menu choose **Mass Properties...**.

50 Mass properties.

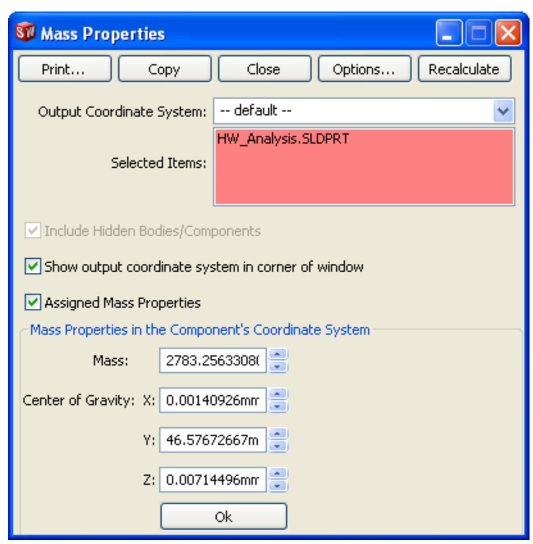
Select the **Mass Properties...** option from the **Tools** menu. The **Density** set with **Edit Materials** is used.

The results of the calculations are displayed in the dialog box.



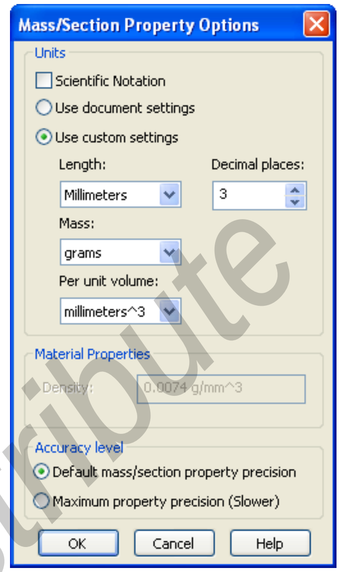
Note

For those parts that do not posses an accurate physical description, you can use **Assigned Mass Properties**. The settings include **Mass** and the location of the **Center of Gravity** (XYZ).



51 Change the settings.
To change the settings, click the **Options...** button and set the **Material Properties**.

This would only change the mass properties for this calculation, not the actual material properties set by the **Material Editor**. Click **Cancel**.

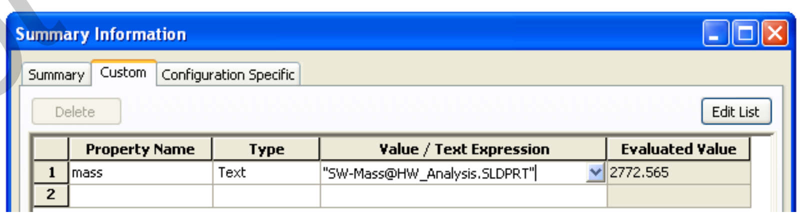


52 Material Editor.
To change the **Material Properties**, use **Edit Material**. See *Edit Material* on page 193.

Mass Properties as Custom Properties

Components of the **Mass Properties** of a part can be carried with the part as a **Custom Property**. This information can be extracted by a Bill of Materials report.

53 File properties.
Click **File, Properties** and click the **Custom** tab. Type in the **Name** `mass`. The **Type** `Text` appears automatically. Assign the mass property component by selecting **Mass** from the **Value/Text Expression** dropdown list. A SolidWorks special property and **Evaluated Value** are created.



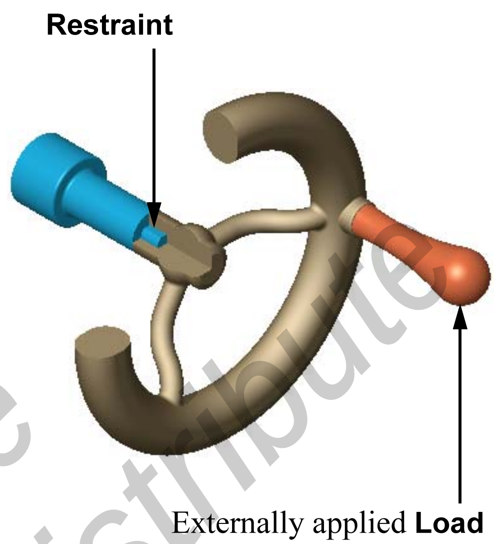
Note

The **Configuration Specific** tab can also be used. This would allow the property to vary by configuration. Configurations will be discussed in *Lesson 9: Configurations of Parts*.

COSMOSXpress

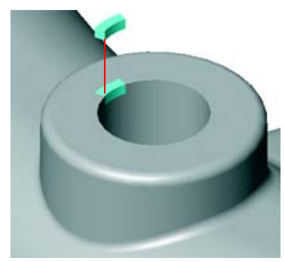
COSMOSXpress is a *first pass* stress analysis tool for SolidWorks users. It helps you judge whether your part will withstand the loading it will receive under real-world conditions. COSMOSXpress is a subset of the COSMOSWorks product.

Overview



COSMOSXpress uses a wizard to provide an easy to use, step-by-step method of performing design analysis. The wizard requires several pieces of information in order to analyze the part: *materials*, *restraints* and *loads*. This information represents the part as it is used. For example, consider what happens when you turn the handwheel. The hub is attached to something that resists turning. This is represented by a *restraint* - the hub is restrained so it doesn't move. A force is applied to the hole in the rim as you attempt to turn the handwheel. This is a *load*. What happens to the spokes? Do they bend? Will they break? This depends on the strength of the material the handwheel is made of, the physical size and shape of the spokes, and the size of the load.

Mesh



In order to analyze the model, COSMOSXpress automatically *meshes* the model, breaking it up into smaller, easier-to-analyze pieces. These pieces are called *elements*.

Although you never see the elements, you can set the coarseness of the mesh prior to the analysis.

Results

The analysis produces results in the forms of **Factor of Safety**, **Stress Distribution** and **Deformed Shape**.

Using the Wizard

The design analysis wizard walks you through the steps of analysis, from **Options** to **Results**. The steps are:

- **Options**
  Setup the type of units that are commonly used for materials, loads and results.
- **Material**
  Choose a material for the part from the standard library or input your own.
- **Restraint**
  Select faces of the part that remain in place (fixed) during the analysis. These are sometime called *constraints*.
- **Load**
  Add external loads such as forces and pressures to induce stress and to deform the part.

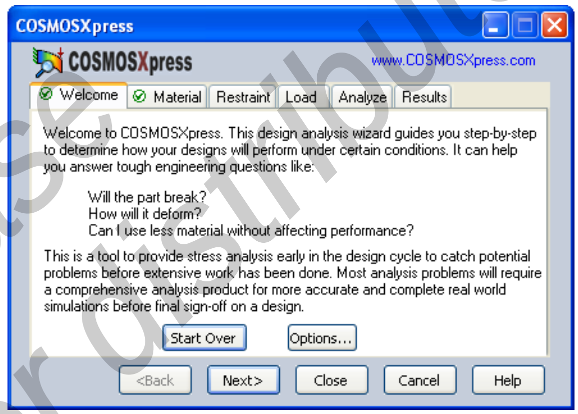
- **Analyze**
  Run the analysis, optionally setting the coarseness of the mesh used.
- **Results**
  View the results of the analysis: Factor of Safety (FOS), Stress and Deformations. This is sometimes called *postprocessing*.

Where to Find It

- From the **Tools** menu, select **COSMOSXpress...**.

---

1 **Start COSMOSXpress.**
Click **Tools, COSMOSXpress...**. The analysis wizard appears.

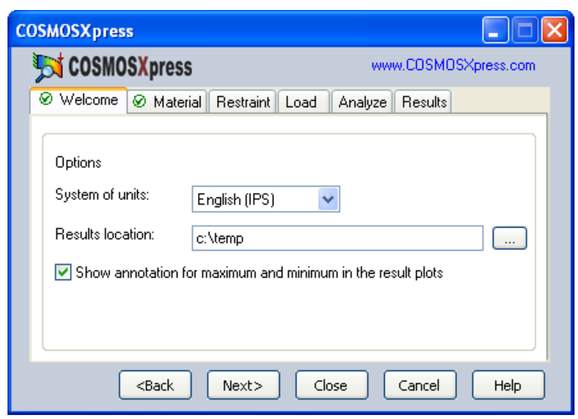


Phase 1: Options

The **Options** dialog contains settings for the **System of units** and **Results location**.

2 **Click Options....**
Set the units to **English (IPS)** and check **Show annotation for maximum and minimum in the stress plot**.

Click **Next**.



Phase 2: Material

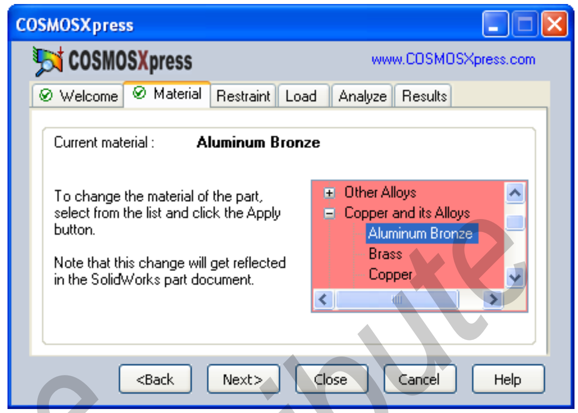
The wizard automatically advances to the next phase as you complete the previous one. As you complete each phase in the wizard, a green check mark is added to the tab.

The next phase is selecting the **Material**. You can choose from libraries of standard materials or add your own.

3 Current material.
The current material, selected within SolidWorks, should be **Aluminum Bronze** from the **Copper and its alloys** list.

To change the material, select it from the list. This is the same list that appears when using **Edit Material**.

Click **Next**.

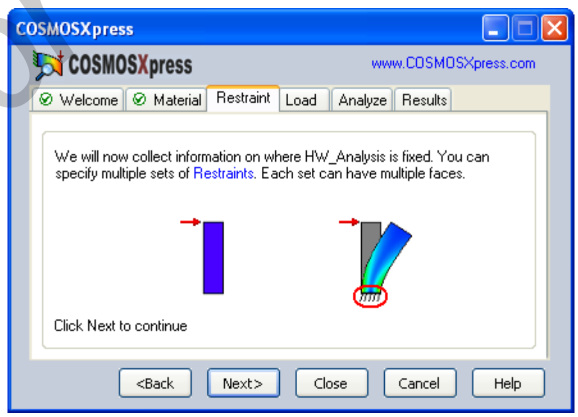


Phase 3: Restraint

Restraints are used to "fix" faces of the model that should not move during the analysis. You must restrain at least one face of the part to avoid analysis failure due to rigid body motion.

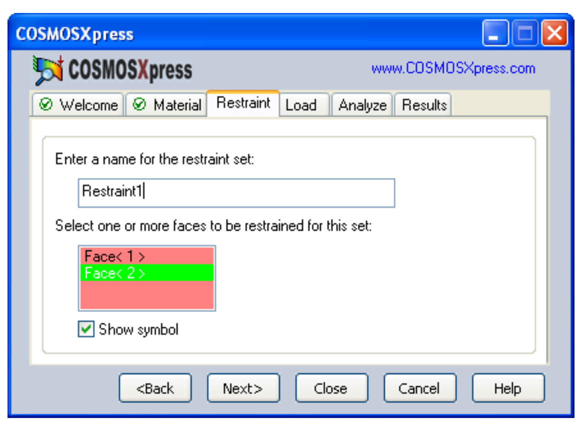
4 Introductory screen.
Click on the blue hyperlinks (such as Restraints) for on-line help.

Click **Next**.

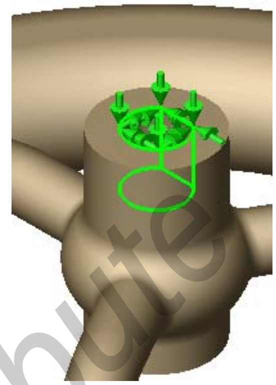


5 Face selection.
Select the cylindrical face and the flat face that form the D-shaped hole.

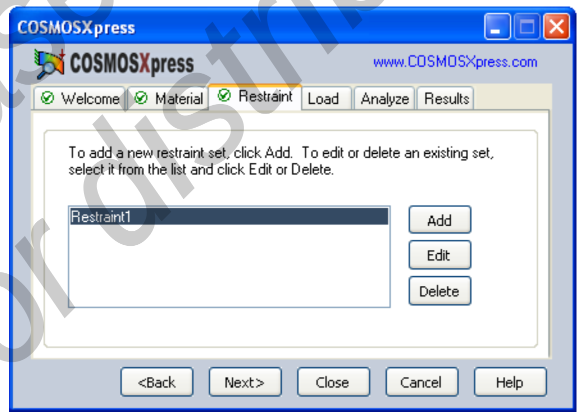
Click **Show symbol** to see a visual display of the restraints.



Click **Next**.



6 Restraint added.
You can **Add**, **Edit**, or **Delete** restraint sets from this menu.



Although COSMOSXpress allows you to create multiple restraint sets, there is little value in doing so because the sets are combined during analysis. In the full COSMOSWorks product, multiple restraint sets are more useful. They allow you to create different analysis cases using different sets of restraints and loads.

Click **Next**.

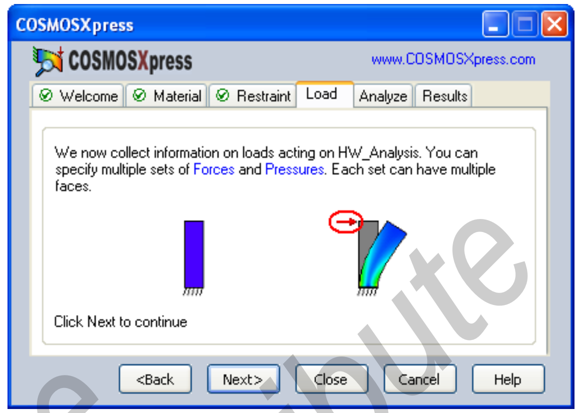
Phase 4: Load

The **Load** tab is used to add external forces and pressures to faces of the part. **Force** implies a total force, for example **200 lbs**, applied to a face in a specific direction. **Pressure** implies that the force is evenly distributed on the face, for example, 300 psi, and is applied normal to the face.

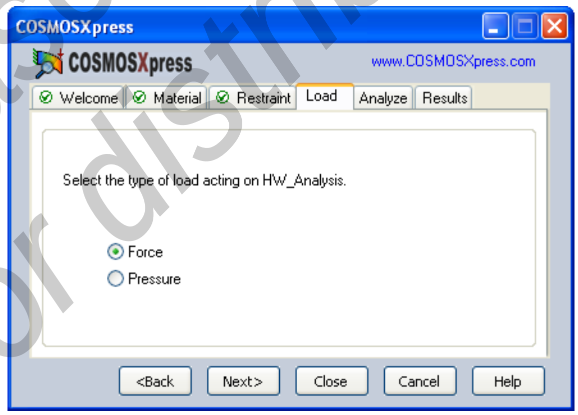
Note

The specified force value is applied to *each* face. For example, if you select 3 faces and specify a 50 lb. force, COSMOSXpress applies a total force of 150 lbs. (50 lbs. on each face).

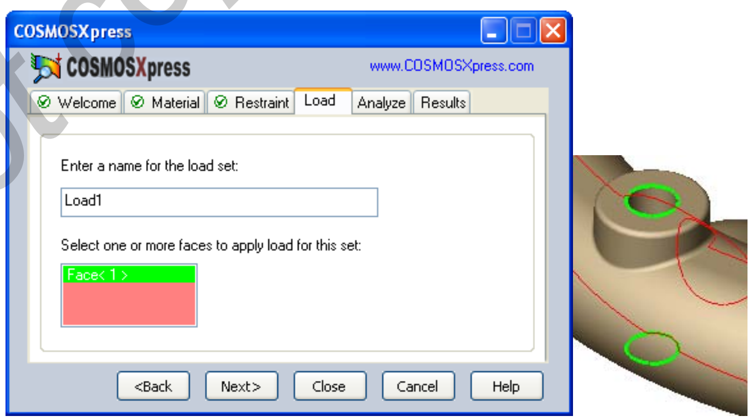
7 Introductory screen.
In this example, we will use a **Force** type load. Click **Next**.



8 Load type.
Click **Force** as the type of load and click **Next**.

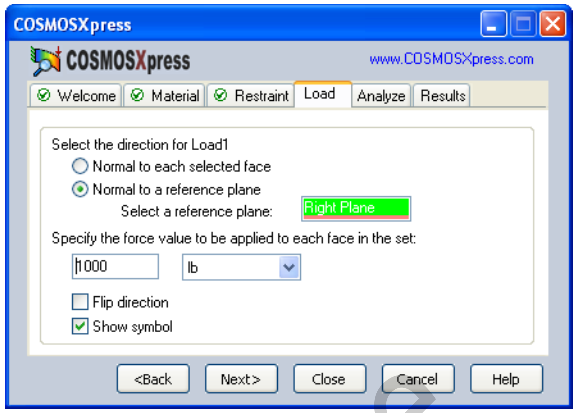


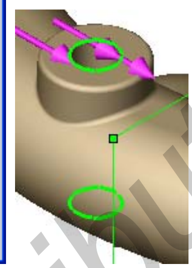
9 Select the face.
Select the cylindrical face as shown and click **Next**.



10 Direction of the force.
Click **Normal to a reference plane** and select the `Right Plane`. Set the value of the force to **1000 lb**.

Click **Show symbol** to make sure that the load is applied in the desired direction. If it is not, click **Flip direction**. In this example the direction really does not matter.



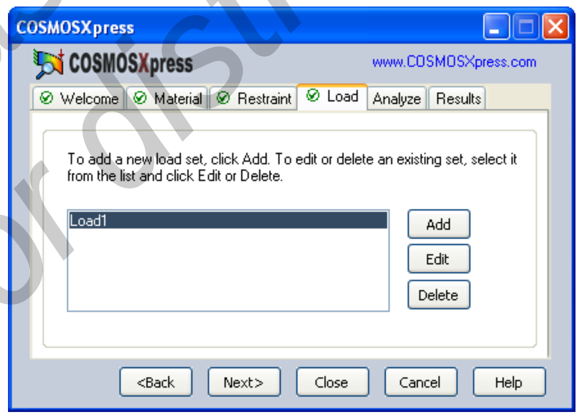


Click **Next**.

11 Completed load set.

The completed load set is listed as `Load1`. Like restraint sets, they can be edited or deleted from this dialog.

Click **Next**.



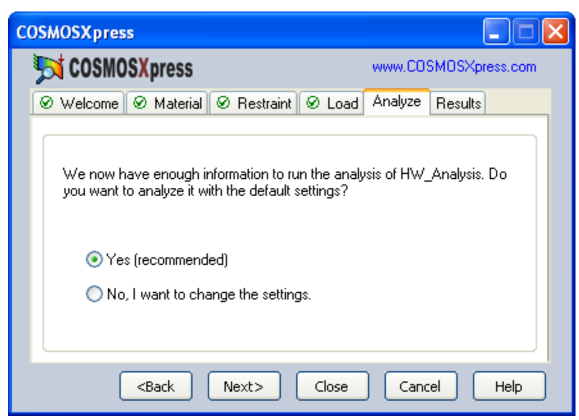
Phase 5: Analyze

COSMOSXpress prepares the model for analysis and then it calculates displacements, strains, and stresses.

12 Analyze screen.

The required information has been provided and the analyzer is ready.

Click **Yes**, and then click **Next**.

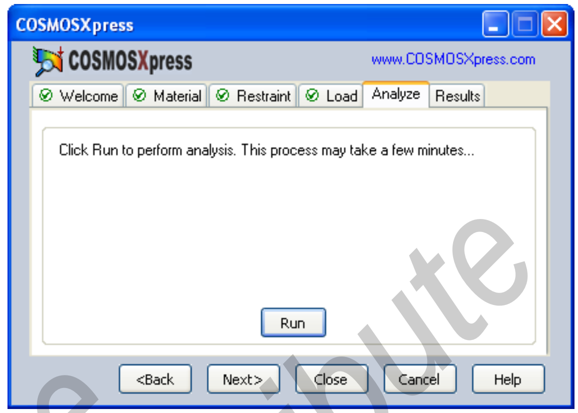


Note

Click **No** if you want to set the size of the elements. Specifying a smaller element size gives more accurate results, but requires more time and resources.

13 Start the analysis.

Click **Run** to begin the analysis. A status window appears. The stages of the analysis process are displayed with elapsed time.



Phase 6: Results

The **Results** tab is used to display the results of analysis. The first result is the **Factor of Safety** (FOS) which compares the yield strength of the material to the actual stresses.

Factor of Safety

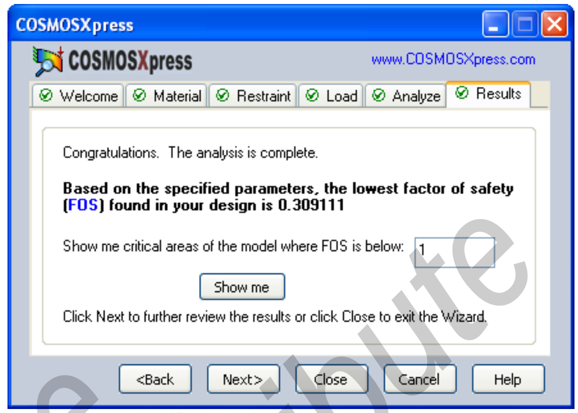
COSMOSXpress uses the maximum von Mises stress criterion to calculate the factor of safety distribution. This criterion states that a ductile material starts to yield when the equivalent stress (von Mises stress) reaches the yield strength of the material. The yield strength (SIGYLD) is defined as a material property. COSMOSXpress calculates the factor of safety at a point by dividing the yield strength by the equivalent stress at that point.

At any location, a factor of safety that is:

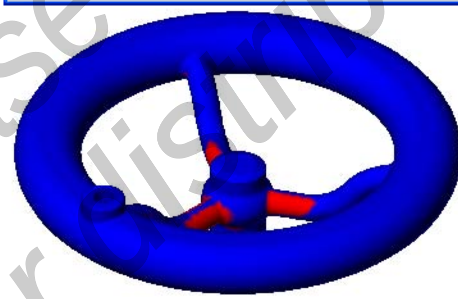
- Less than 1.0 indicates that the material at that location has yielded and that the design is not safe.
- Equal to 1.0 indicates that the material at that location has just started to yield.
- Greater than 1.0 indicates that the material at that location has not yielded.

14 Factor of safety.

The **FOS** is listed as less than **1**. This indicates that areas of the part are overstressed and will fail.



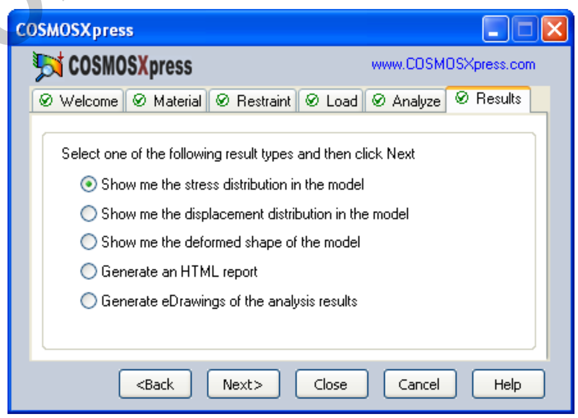
Click **Show Me** to display a colored image representing the factor of safety. Red areas indicate where the factor of safety is less than one.



Click **Next**.

15 Result types.

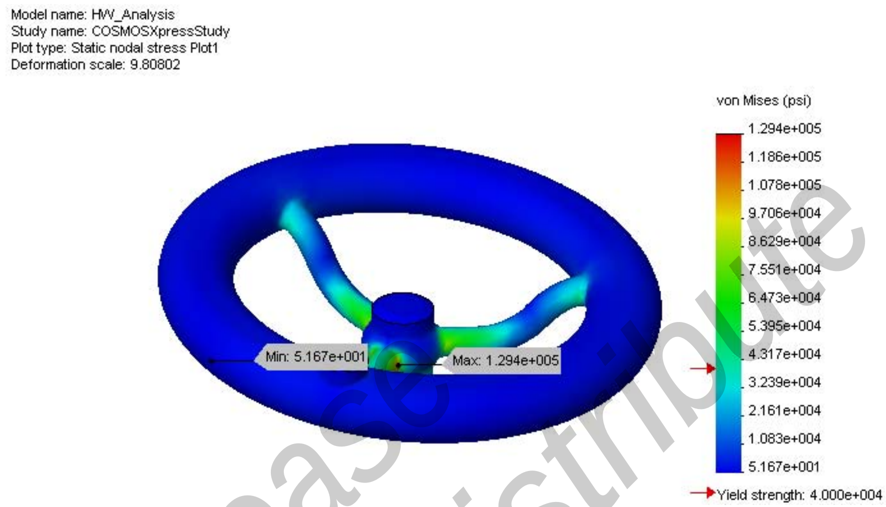
There are other ways to look at the results: stress, displacement and deformation.



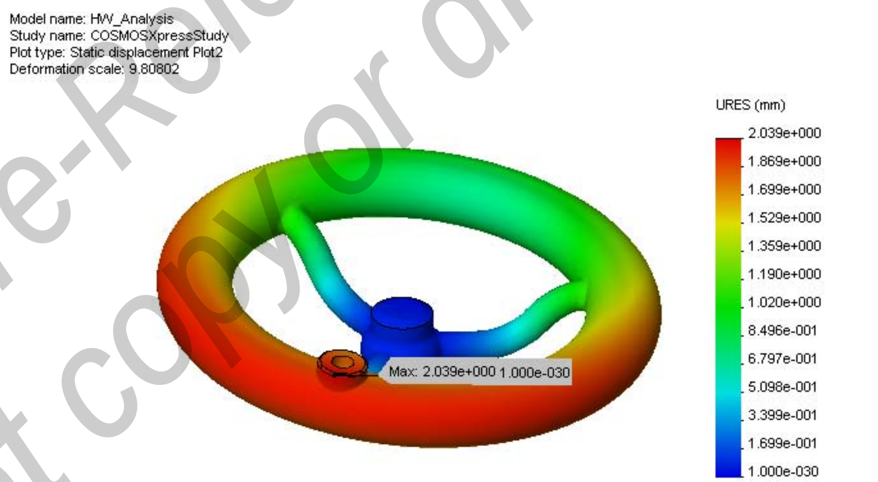
The following are some examples of the different ways to display the results. The **Stress Distribution** and **Deformed Shape** graphics can be animated and saved as `*.avi` files. Your instructor will demonstrate these animations in class.

Note The displays are exaggerated by the Deformation Scale.

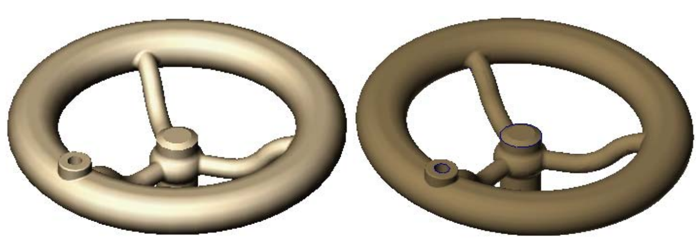
- **Stress Distribution**



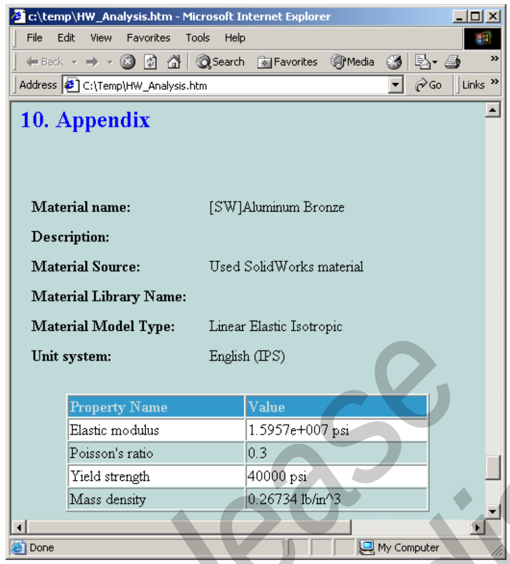
- **Displacement Distribution**



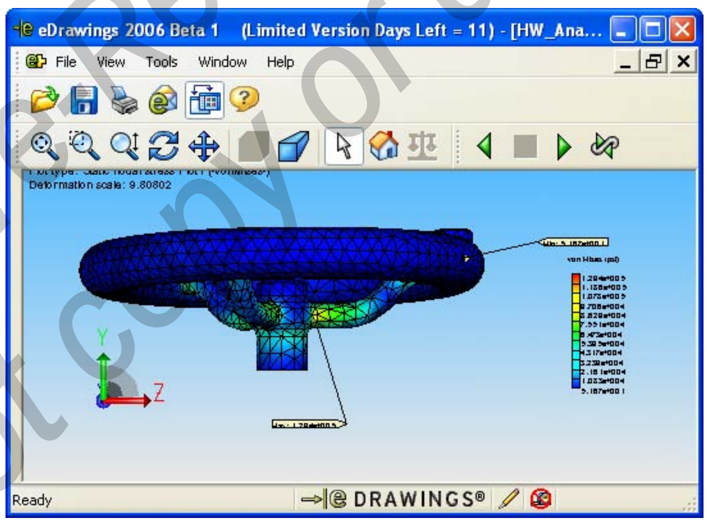
- **Deformed Shape**
This is best seen by viewing the animation.



- **HTML Report**



- **eDrawing of analysis results**



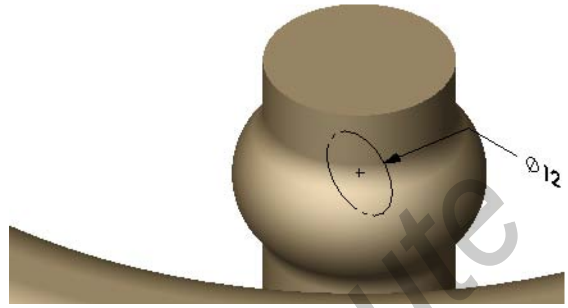
16 Close and Save.

Click the **Close** button. You will be asked if you want to save the COSMOSXpress data. Click **Yes**.

Updating the Model

Changes performed in SolidWorks are detected by COSMOSXpress. Changes can be made to the model, materials, restraints or loads. The existing analysis can be **Updated** to show the newest results.

17 Edit profile sketch.
Expand the `Spoke` feature and edit the profile sketch. Select the circle and click **For construction**.



Introducing: Insert Ellipse

Sketching an ellipse is similar to sketching a circle. Position the cursor where you want the center and drag the mouse to establish the length of the major axis. Then release the mouse button. Next, drag the outline of the ellipse to establish the length of the minor axis.

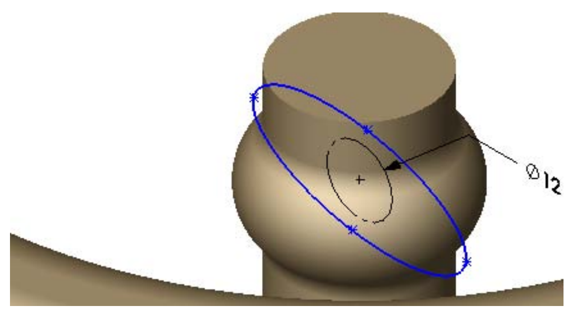
Important!

To fully define an ellipse you must dimension or otherwise constrain the lengths of the major and minor axes. You must *also* constrain the orientation of one of the two axes. One way to do this is with a **Horizontal** relation between the ellipse center and the end of the major axis.

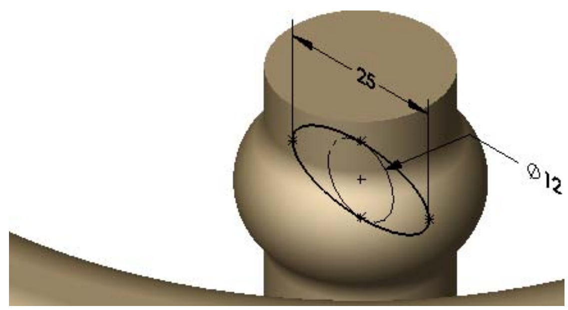
Where to Find It

- Click **Tools, Sketch Entity, Ellipse**.
- Or, click **Ellipse** on the Sketch Tools toolbar.

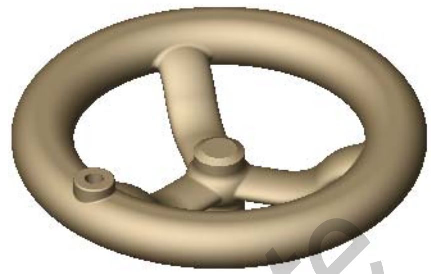
18 Ellipse.
Click **Ellipse** and position the centerpoint at the centerpoint of the circle. Move away from center and position the major and minor axes with additional clicks.



19 Relations and dimensions.
Add relations to make the major axis points Horizontal and one of the minor axis points Coincident to the circle. Add the dimension as shown.

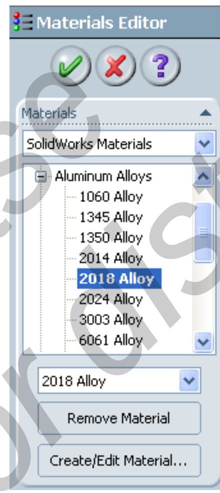


20 Rebuild the part.
The edited sweep profile is used with the same path to produce a different result.



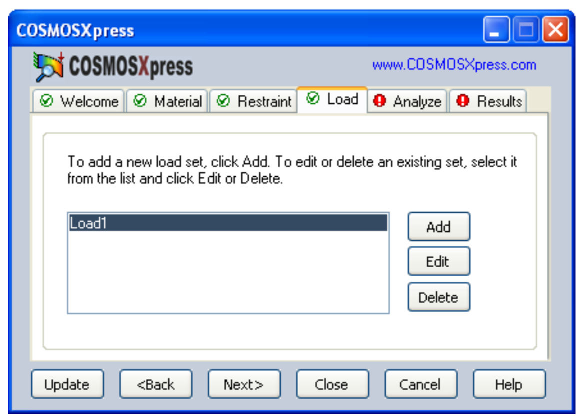
21 Change material.
Change the material using right-click **Edit Material**. Select **Aluminum Alloys, 2018 Alloy**.

The **Visual Properties** of the material are also applied by default. If you are using **Advanced graphics** and **RealView**, the part will appear similar to the one shown here.



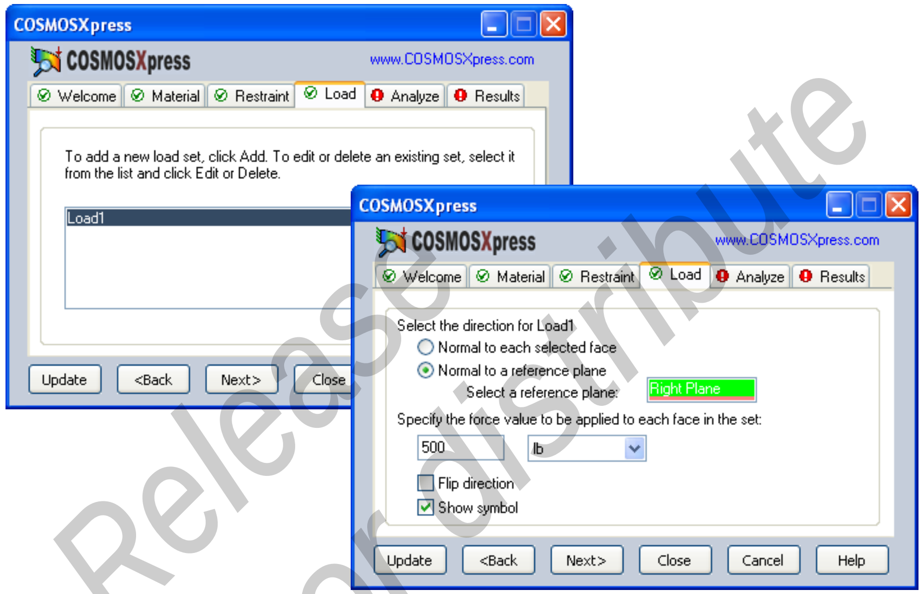
22 Errors.
Start COSMOSXpress again and note the changes to the wizard. The **Analyze** and **Results** tabs have error markers.

Other changes (loads, restraints) can also be made at this point, prior to reanalyzing the part.



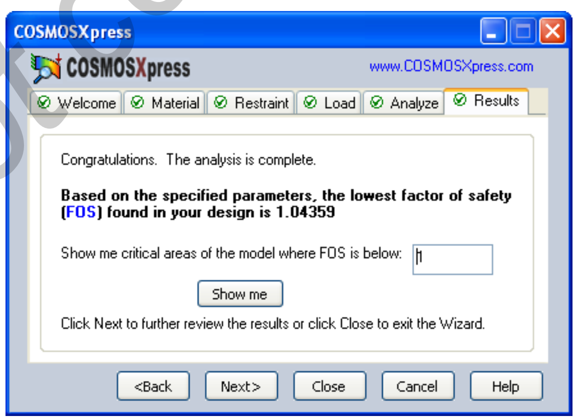
23 Change load value.
Click the **Load** tab and click **Edit**.

Click **Next** and change the force value to **500**.



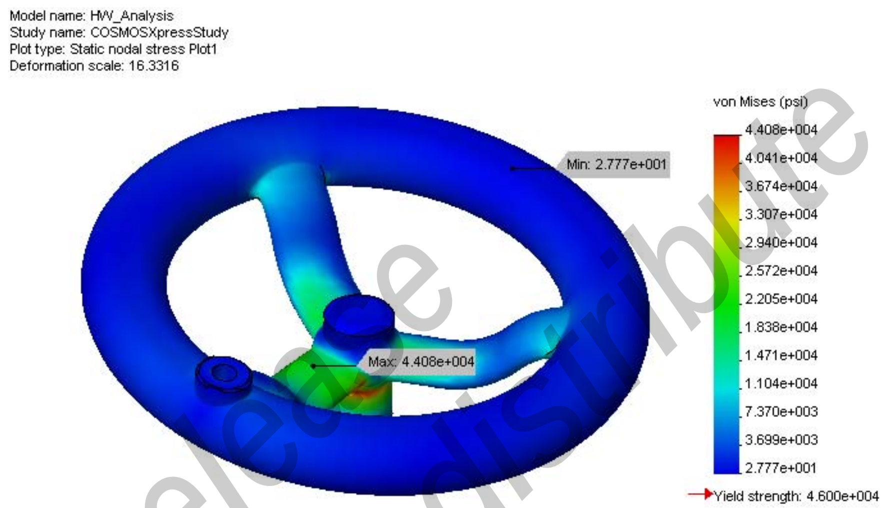
24 Update.
Click **Next** and **Update**.

25 Results.
The factor of safety has increased with the change in geometry. The part is no longer overstressed.



26 Reduced stress.

Along with the increase in the factor of safety comes the expected decrease in stress. The deformations are also smaller.



Exercise 21: Flange

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- Revolved features.
- Circular patterning.

Units: **inches**

Design Intent

The design intent for this part is as follows:

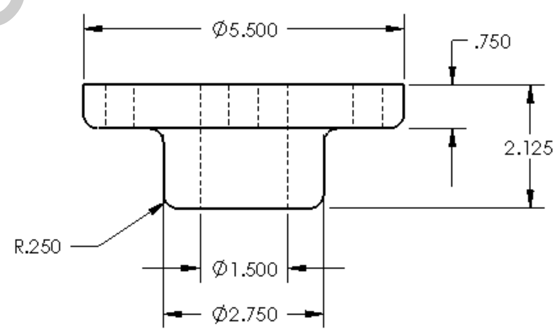
1. Holes in the pattern are equally spaced.
2. Holes are equal diameter.
3. All fillets are equal and are **R0.25”**.

Note that construction circles can be created using the **Properties** of a circle.

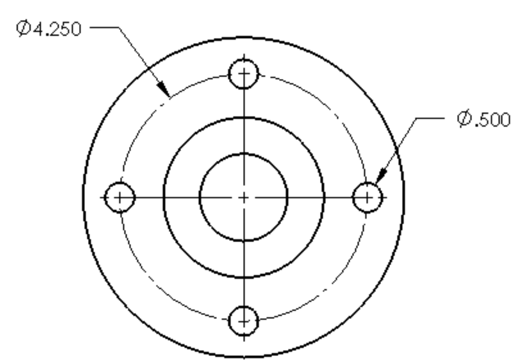
Dimensioned Views

Use the following graphics with the description of the design intent to create the part.

Top View



Front View



Exercise 22: Wheel

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- Revolved features.
- Optional: Text in a sketch.

Units: **millimeters**

Design Intent

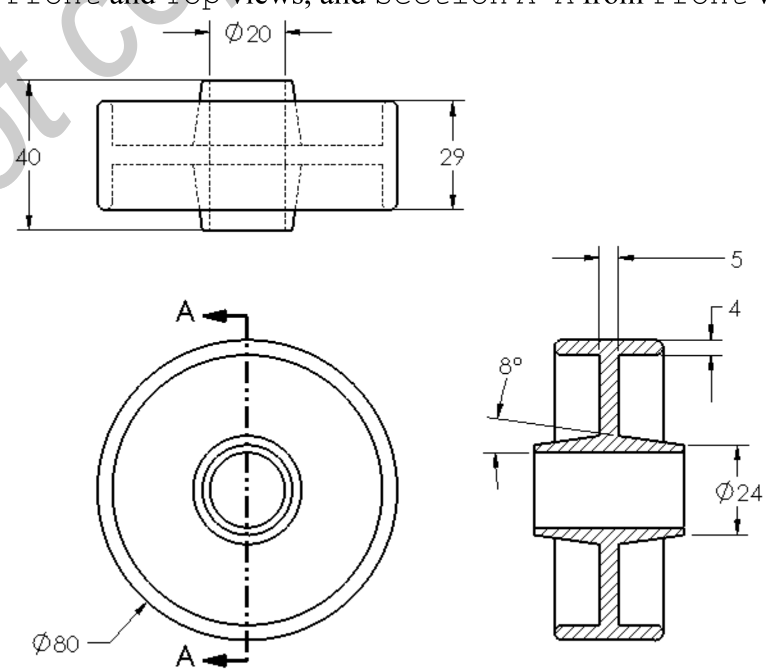
The design intent for this part is as follows:

1. Part is symmetrical about the axis of the hub.
2. Hub has draft.

Dimensioned Views

Use the following graphics with the description of the design intent to create the part.

`Front` and `Top` views, and `Section A-A` from `Front` view.



SECTION A-A

Optional: Text in a Sketch

Text can be added to a sketch and extruded to form a cut or a boss. The text can be positioned freely, located using dimensions or geometric relations, or made to follow sketch geometry or model edges.

Introducing: Text Tool

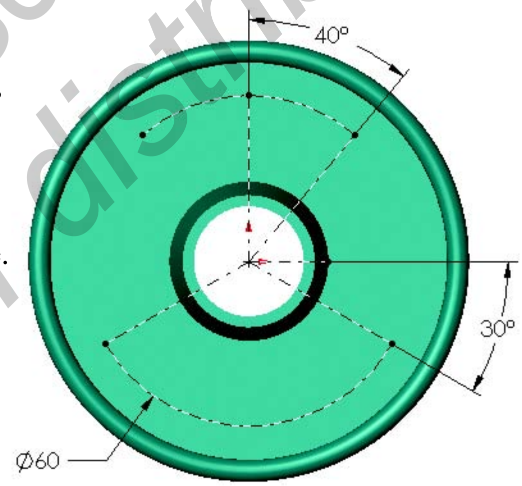
The text tool allows you to insert text into a sketch and use it to create an extruded boss or cut feature. Since SolidWorks software is a true Windows application, it supports whatever fonts you have installed on your system.

Where to Find It

- Click **Tools, Sketch Entities, Text...**.
- Or, on the Sketch toolbar click **Text**.

---

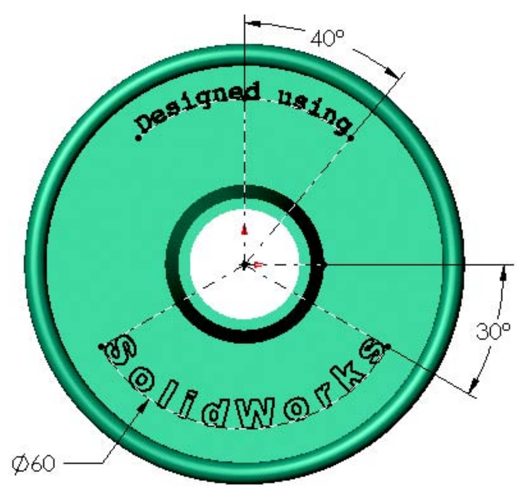
1 **Construction geometry.**
Sketch on the front face and add construction lines and arcs as shown.



Tip

Use **Symmetric** relationships between the endpoints of the arcs and the vertical centerline.

2 **Text on a curve.**
Create two pieces of text, one attached to each arc. They have the following properties:



- **Text: Designed using**
- **Font: Courier New 11pt**
- **Alignment: Center Align**
- **Width Factor: 100%**
- **Spacing: 100%**

- **Text: SolidWorks**
- **Font: Arial Black 20pt.**
- **Alignment: Full Justify**
- **Width Factor: 100%**
- **Spacing: not applicable when using Full Justify**

3 **Extrude.**
Extrude a boss with a **Depth** of **1mm** and **Draft** of **1°**.

4 **Save the part and close it.**

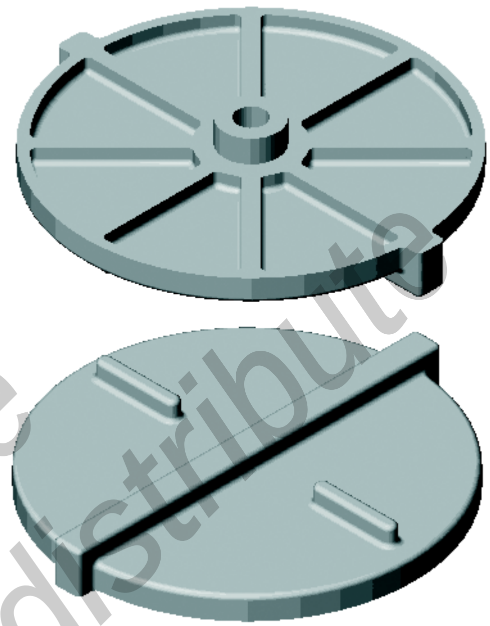
Exercise 23: Compression Plate

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- Sketching.
- Revolved features.
- Symmetry.

Units: **millimeters**



Design Intent

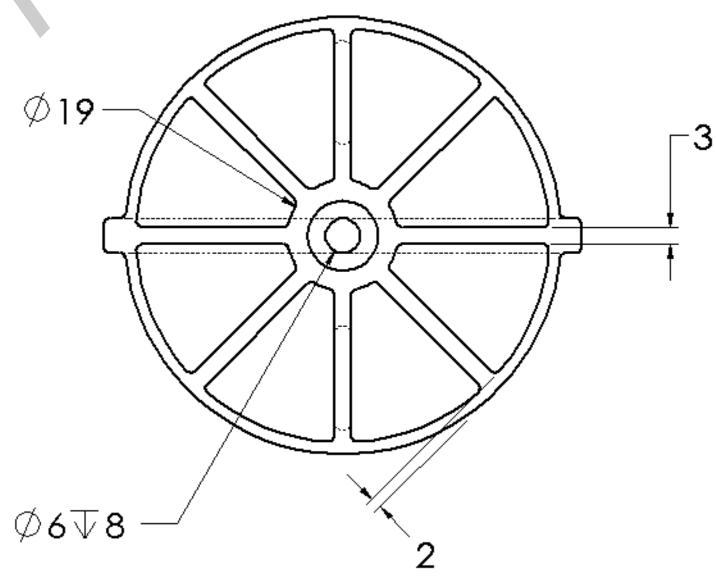
The design intent for this part is as follows:

1. Part is symmetrical.
2. Ribs are equally spaced.
3. All fillets and rounds are **1mm**.

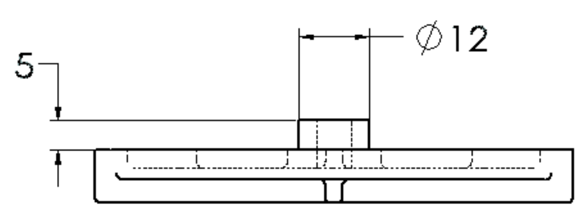
Dimensioned Views

Use the following graphics with the design intent to create the part.

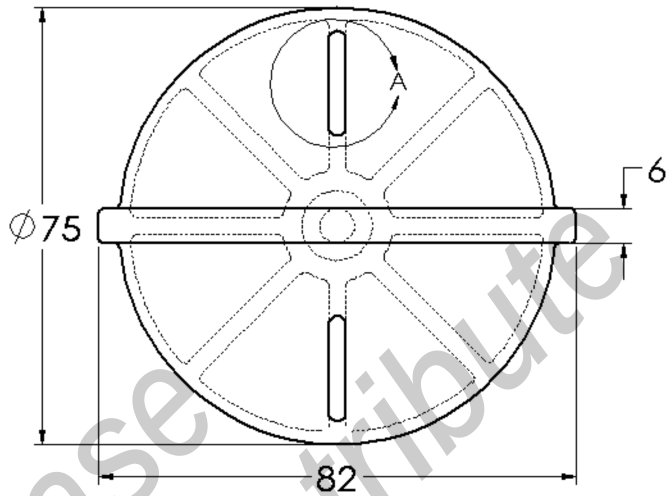
Top view



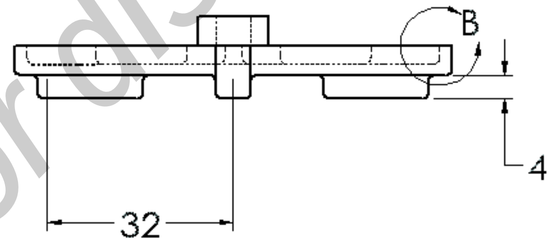
Front view



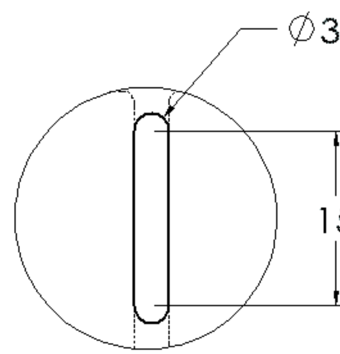
Bottom view



Right view

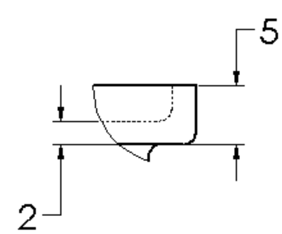


Detail A



DETAIL A
SCALE 4 : 1

Detail B



DETAIL B
SCALE 4 : 1

Exercise 24: Tool Post

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- Sketching.
- Revolved features.
- Sketch offsets.

Units: **inches**

Design Intent

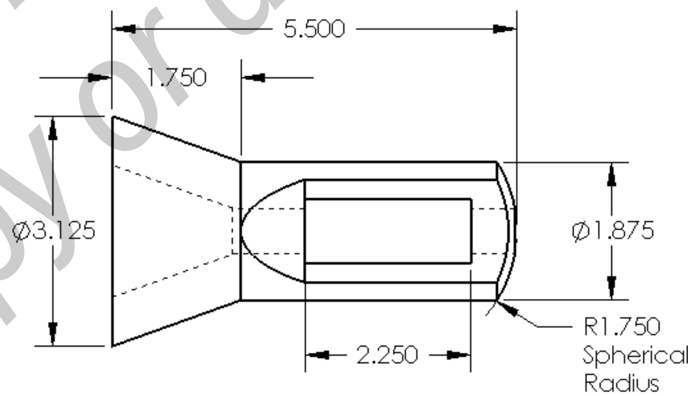
The design intent for this part is as follows:

1. Part is symmetrical.
2. Center hole is though all.

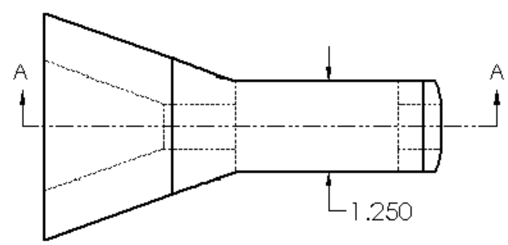
Dimensioned Views

Use the following graphics with the design intent to create the part.

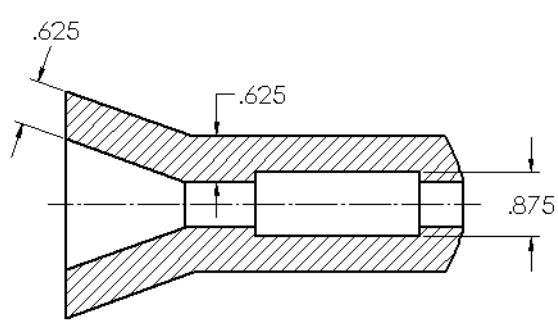
`Top` view



`Front` view



`Section A-A` from `Front` view



SECTION A-A

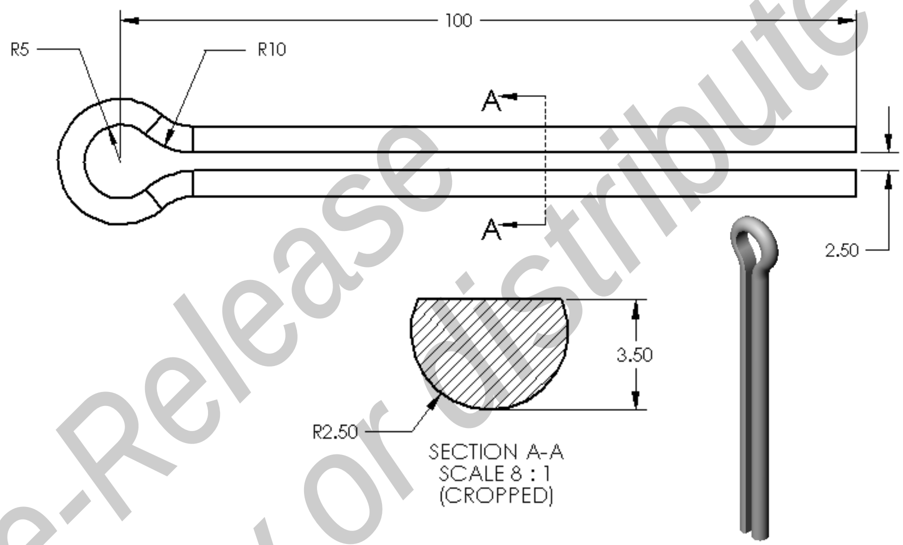
Exercise 25: Sweeps

Create these three parts using swept features. These require a path and a section.

Units: **millimeters**

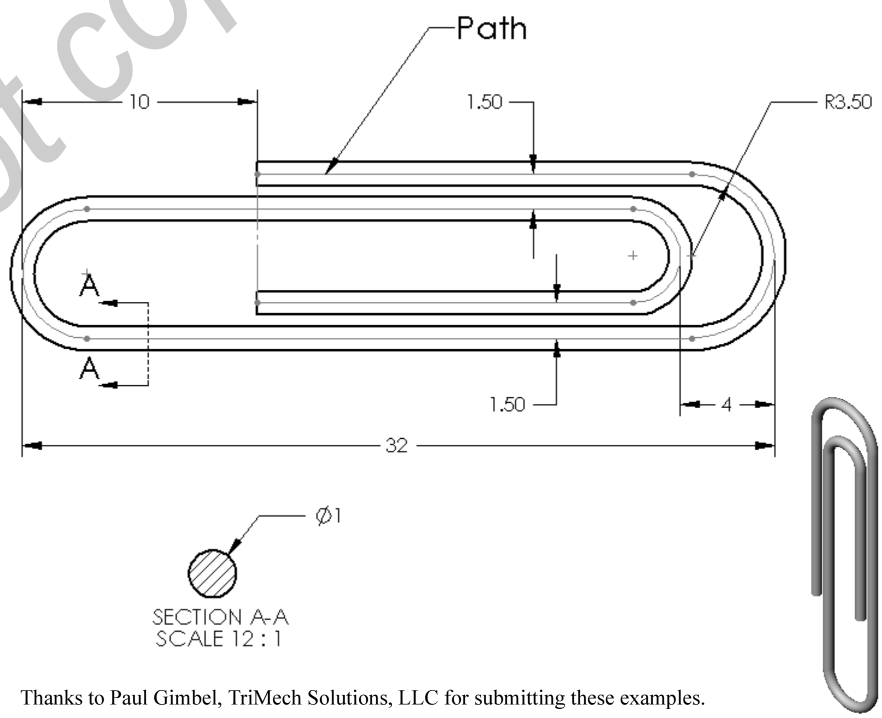
Cotter Pin

The `Cotter Pin` uses a path that describes the inner edge of the sweep.



Paper Clip

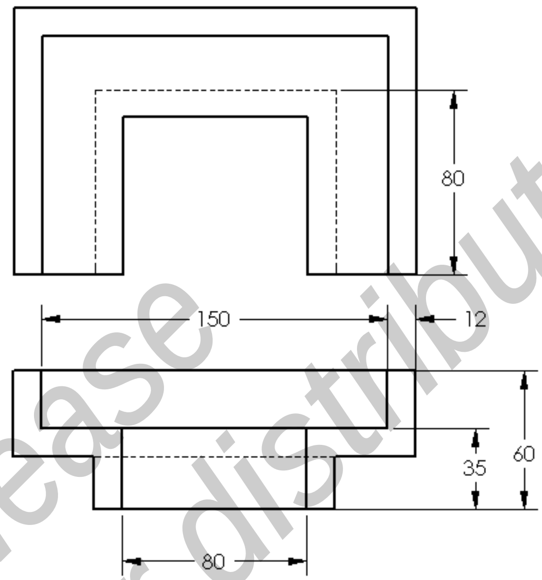
The `Paper Clip` is defined by a path that describes the centerline of the sweep.

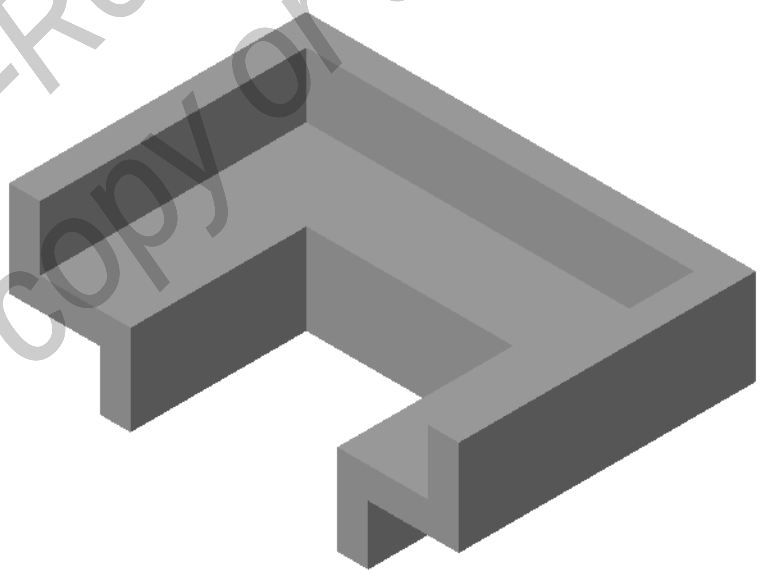


Thanks to Paul Gimbel, TriMech Solutions, LLC for submitting these examples.

Mitered Sweep

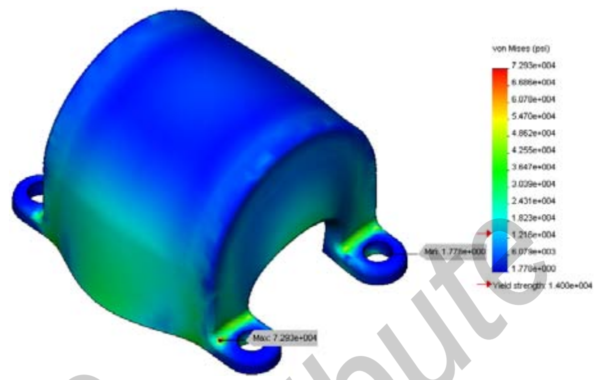
The `Mitered Sweep` is defined by a path that describes the outer edge of the sweep.





Exercise 26: COSMOSXpress

Perform a first pass stress analysis on an existing part.



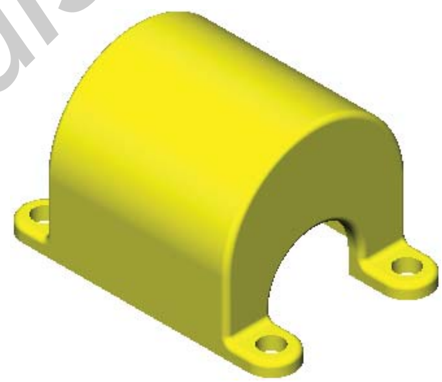
This lab uses the following COSMOSXpress skills:

- Assigning material properties.
- Defining restraints.
- Defining loads.
- Running an analysis.
- Displaying the results.

Units: **inches**

1 **Open `Pump Cover`.**
This part represents a cover that will be filled with oil under high pressure.

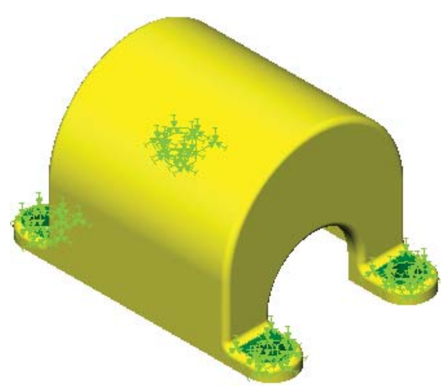
Start the COSMOSXpress wizard.



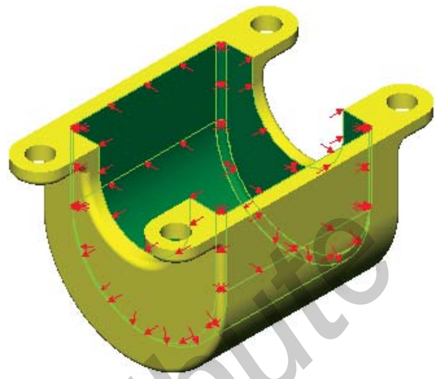
2 **Set the units.**
Click **Options...** and set the units to **English (IPS)** and check **Show annotation for maximum and minimum in the result plots**. Click **Next**.

3 **Specify the material.**
Select **Aluminum Alloys** and select **2014 Alloy** from the list.

4 **Define the restraint set.**
Select the uppermost faces of the four tabs and the cylindrical faces of the four bolt holes.



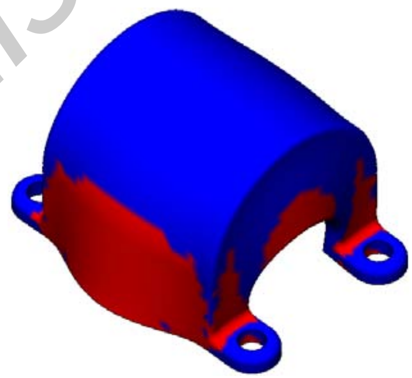
5 Define the load set.
Select **Pressure** for the type of load. Right-click one of the faces on the *inside* of the `Pump Cover`. Pick **Select Tangency** from the shortcut menu.



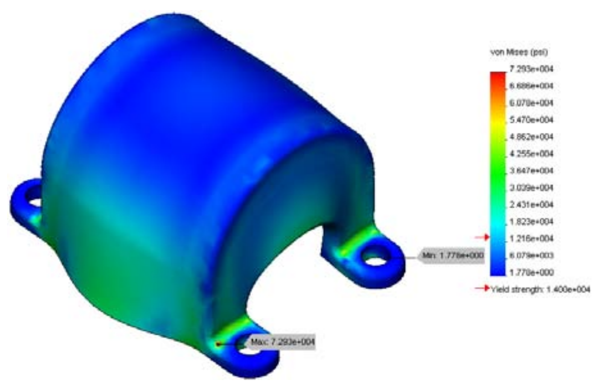
6 Set the pressure value and direction.
Set the pressure value to **500 psi**. Click **Show symbol** and verify that the arrows are pointing in the correct direction.

7 Run the analysis.
Use the default mesh settings.

8 Results.
The factor of safety is less than 1 indicating that the part is over stressed.



9 Stress distribution and deformation.
Display the stress distribution in the model. Play the animation of the deformation.



10 Change the material.
Change the material to **Ductile Iron**.

11 Update.
Click **Update** to rerun the analysis using the new material.

12 Factor of Safety.
The new factor of safety is in excess of 1.

13 Close and save.
Click **Close**. Click **Yes** to save the COSMOSXpress data.

14 Save and close the part.

Lesson 7
Editing: Repairs

Upon successful completion of this lesson, you will be able to:

- Diagnose various problems in a part and repair them.
- Utilize all the available tools to edit and make changes to a part.

Part Editing

The SolidWorks software provides the capability to edit virtually anything at any time. In order to emphasize this, the major tools for editing parts are covered and reviewed here in one lesson.

Stages in the Process

Some key stages in the process of modifying this part are shown in the following list. Each of these topics comprises a section in the lesson.

- **Add and delete relations**
  Sometimes the relations in a sketch must be deleted or changed due to changes in the design.
- **What's Wrong?**
  When errors occur, the **What's Wrong** option can be used to investigate and pinpoint the problem.
- **Edit sketch**
  Making changes to the geometry and relations of any sketch can be done through **Edit Sketch**.
- **Check sketch for feature**
  Check Sketch can check a sketch for problems, verifying its suitability for use in a feature. You must **Edit Sketch** before using **Check Sketch for Feature**.
- **Edit feature**
  Changes to how a feature is created are done through **Edit Feature**. The same dialog that is used to create a feature is used to edit it.
- **Edit sketch plane**
  Sketched on `Front` instead of `Top`? Use **Edit Sketch Plane** to transfer the sketch from the current plane to a different plane or face.
- **Reorder**
  Features that have been created in the wrong order can be reordered by simple dragging in the FeatureManager design tree.
- **Rollback**
  Rollback and roll forward are used to visit previous stages of the model. This allows you to see the model in earlier versions and add in missing features.
- **Change dimension value**
  This is probably one of the most typical changes. If design intent was properly captured, changes in dimensions cause changes in the size of individual features and ultimately the entire model.

Editing Topics

Editing covers a wide range of topics from fixing broken sketches to reordering things in the FeatureManager design tree. These topics can be summarized as repairing errors, interrogating the part, and changing the design of the part. Each is described below.

Information from a Model

Nondestructive testing of a model can yield many important insights as to how the model was created, the relationships that were established, and changes that can be incorporated. This section will focus on using editing tools in conjunction with rollback to "interrogate" the model.

Finding and Repairing Problems

Finding and repairing problems in a part is a key skill in solid modeling. Many changes that are made to a given part (**Edit Feature**, **Edit Sketch** and **Reorder**, to name a few) can cause features down the line to fail. Pinpointing the problem area and finding the solution will be discussed in this section.

Problems can occur in sketches or any other feature of the part. Although there are many types of errors, there are some that occur more often than others. Dangling dimensions and relations are very common, as is extraneous geometry in sketches. Using some of the tools available in the SolidWorks software, problems can be easily diagnosed and repaired.

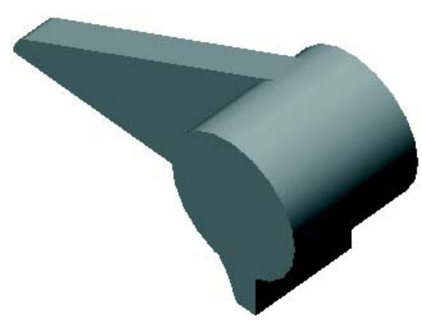
Opening a part that has errors can be confusing. One error near the beginning of the process can often cause many later features to fail along with it. Repairing that initial error may fix the rest of the errors as well. Some repairs will be made to this model *before* interrogating and changing it.

Procedure

We will begin by opening an existing part.

1 **Open the part named `Editing CS`.**
This part was built and saved with numerous errors.

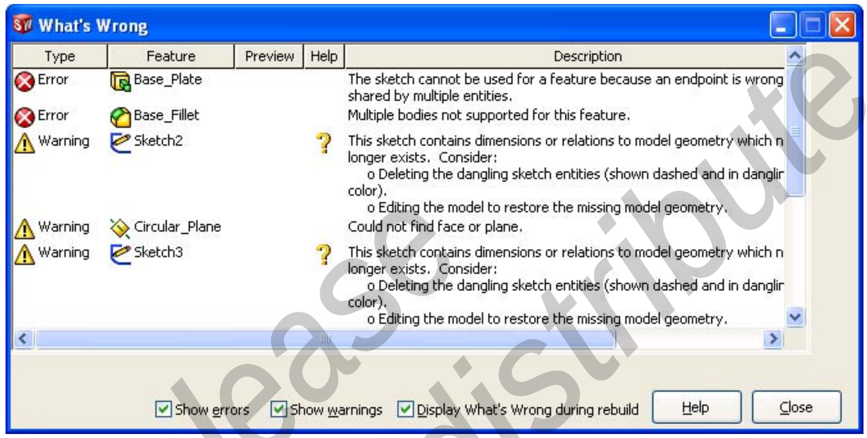
2 **Rebuild errors.**
Upon opening, the system displays a message box, labelled with the part name and the phrase **What's Wrong**. Each error is listed by feature name in the scrollable dialog.



Only a portion of the model is visible; errors have caused some features to fail.

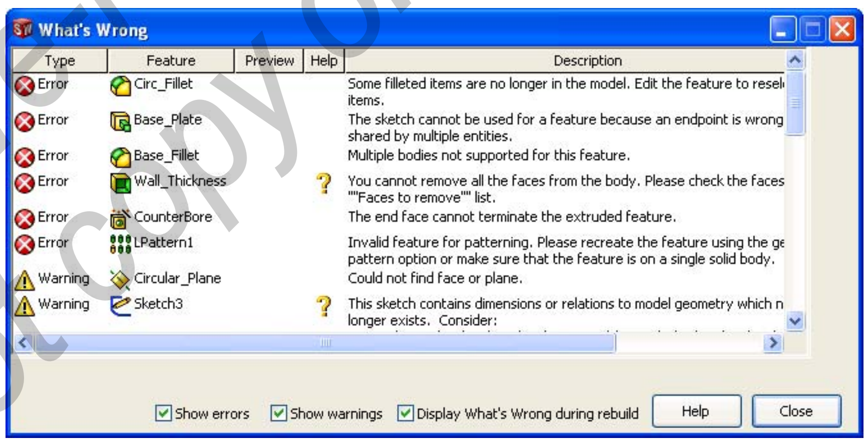
What's Wrong Dialog

The **What's Wrong** dialog lists all the errors in the part. The errors themselves are broken down into **Errors** that prevent features from being created and **Warnings** that do not. The other columns offer some help in diagnosing the problem including a preview in some cases.



Tip

The columns of the dialog can be sorted by the column headers. Click on the **Type** header to sort by **Error** and **Warning** types.



Tip

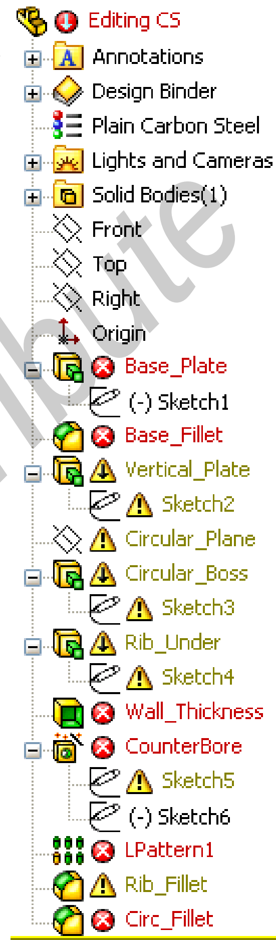
Click the question mark **?** to open on-line help regarding this type of error.

Note

The display of this error dialog is controlled by the option **Show errors every rebuild** on the **Tools, Options, System Options, General** menu. This option must be *enabled* in order for this message to appear. There are several controls:

- Through the **Tools, Options...** dialog
- Through the message dialog itself: **Display What's Wrong during rebuild**
- Through the message dialog itself: display of just errors (**Show errors**), just warnings (**Show warnings**) or both

3 FeatureManager design tree.
The FeatureManager design tree lists many errors indicated with markers. The markers placed next to the features have particular meanings:



- **Top Level Error**

The **Top Level Error** marker marks an error in the tree below. Useful in assemblies and drawings to see part errors.

- **Expand**

An **Expand** marker is placed next to a feature that has an error or warning on the feature beneath it. Expand the feature to see the problem. The text of the feature is shown in *green*.

- **Error**

An **Error** marker is placed next to a feature that has a problem and *cannot* create geometry. The text of the feature is shown in *red*.

- **Warning**

A **Warning** marker is placed next to a feature that has a problem but creates geometry. This is common for “dangling” geometry and relations. The text of the feature is shown in *green*.

- **Normal Features**

Normal features that do not have warnings or errors appear with *black* text.

Where to Begin

Features are rebuilt in sequence from the top of the tree. The best place to begin is at the first (base) feature with an error, in this case that is the feature `Base_Plate`. An error in the base feature may cause a series of errors in the child features.

4 What's Wrong?

The **What's Wrong** option is used to highlight an error message for a selected feature. Right-click the `Base_Plate` feature and select **What's Wrong?**. The message indicates that the sketch cannot be used for the feature because an endpoint is wrongly shared.

5 Edit the sketch.

The **What's Wrong** message has indicated the sketch (`Sketch1`) as the problem. Edit the sketch of the feature.

Tip

For Over Defined sketch problems, see *Overdefined Sketches* on page 114 of *Modeling a Casting or Forging* in *Volume 1*.

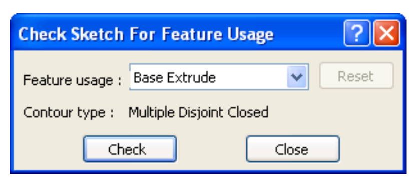
Check Sketch for Feature

Check Sketch for Feature allows you to check the validity of a sketch for use in a feature. This can be done either before the feature is created or, as in this example, after. Since different features have different sketch requirements – for example, revolved features require an axis of revolution – you select the type of feature for which the sketch is to be evaluated. Any geometry that impedes the creation of that feature will be highlighted. It will also check for missing and inappropriate geometry.

Where to Find It

- From the **Tools** menu, select **Sketch Tools, Check Sketch for Feature...**.

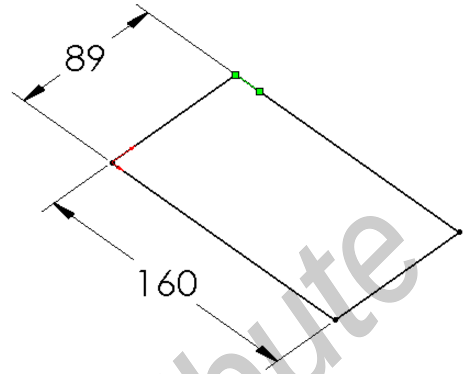
6 Check Sketch.



The **Check Sketch for Feature...** command checks for incorrect geometry in the sketch, compared to what is required by the **Contour type**. In this case the **Feature Usage** is set to **Base Extrude** because that is the type of feature this sketch belongs to. The **Contour type** is determined from the type of feature.

Click **Check**.

7 Message.
A message appears stating the same message that we got from **What's Wrong** and the system highlights the problem geometry, in this case, one of the three lines that share a common endpoint.

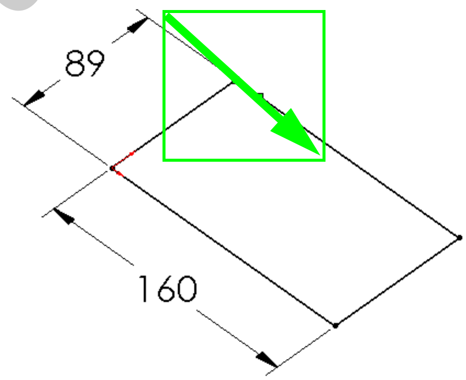


Click **OK** to close the message window and click **Close** to close the **Check Sketch for Feature** dialog box.

Box Selection

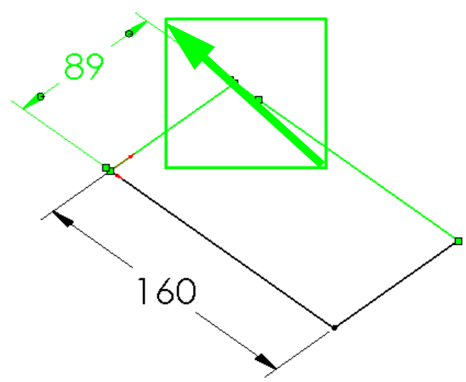
Box selection allows you to select multiple sketch entities with a drag-window. Entities are selected based on whether the window is dragged from right to left or left to right. The selection includes dimensions.

Left to right: Only the geometry completely within the window (the short line) is selected.



Right to left: The geometry within and crossing the window (short line and two long lines) is selected.

This is also called **cross selection**.



Tip

Using **Shift** with the box selection maintains any previous selections. Using **Control** with the box selection clears previous selections.

These selection methods will not be used in this example. Instead, an automated repair method will be used.

Repairing the Sketch

Making repairs to a sketch can be accomplished in several ways. In a simple case like this one, a single line is causing the error. That line was highlighted by **Check Sketch for Feature**. The line can simply be deleted and the **Check Sketch for Feature** command can be used again to confirm that the sketch is error free.

A more automated method using **Repair Sketch** can be used to fix the errors.

Introducing: Repair Sketch

Repair Sketch is used to analyze errors in a sketch and repair them. It can be used to repair errors that stem from small gaps, overlapping geometry and multiple short segments.

Where to Find It

- From the **Tools** menu, select **Sketch Tools, Repair Sketch**.
- Or, on the 2D to 3D toolbar click the **Repair Sketch** tool.

---

8 **Repair.**
Click **Tools, Sketch Tools, Repair Sketch**. The system deletes the offending line.

9 **Check again.**
Use **Check Sketch for Feature** again to see the result of the repair. The dialog displays no problems with the sketch.

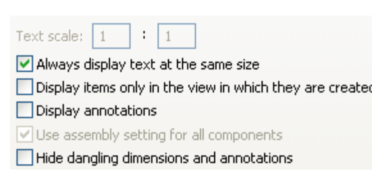
10 **Add dimensions and relations.**
Dimensions and relations can be deleted during the process of repairing a sketch. Add replacement dimensions and relations and exit the sketch.

11 **Remaining errors.**
All the errors and warnings that remain are listed in the dialog. So that you don't see this message dialog every time you make a correction, deselect the **Display What's Wrong during rebuild** option. More of the model appears.

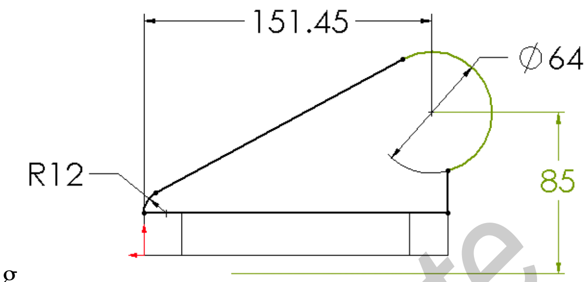
12 **Next error.**
The top error on the list is for `Sketch2` under the feature `Vertical_Plate`. It contains **Dangling sketch entities** according to the message. Dangling sketch entities are found when dimensions or relations reference things that no longer exist.

Note

Dangling dimensions and relations can be hidden from view. The **Hide dangling dimensions and annotations** option can be found under **Tools, Options, Document Properties, Annotations Display**.



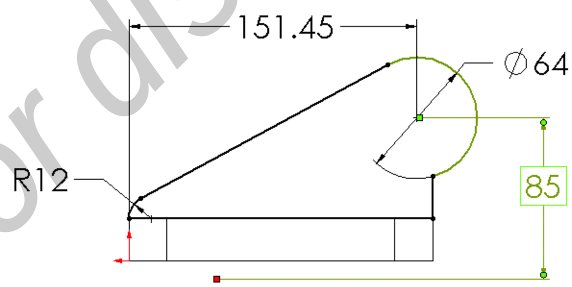
13 Edit the sketch.
Edit the sketch of the `Vertical_Plate` feature. Note that the **85mm** dimension is a different color than the other dimensions. This is the color used for dangling dimensions and relations. The dimension is trying to attach to geometry that no longer exists, and therefore it is considered dangling.



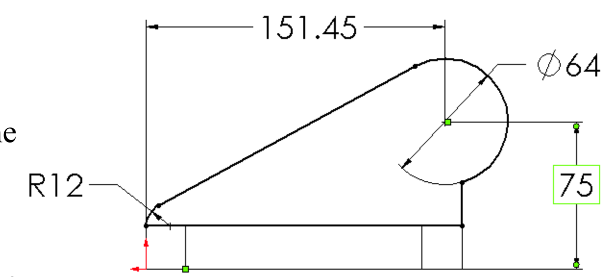
Reattach Dimensions

Dangling dimensions and relations can be quickly repaired by reattching them to the model.

14 Select the dimension.
Click on the **85mm** dimension to see the drag handles. The end marked red is the dangling end. Dangling relations on geometry are marked in a similar fashion.



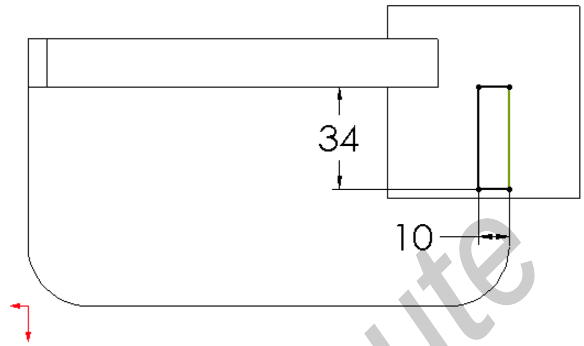
15 Drag and drop.
Drag the red handle and drop it on the bottom edge of the part when the edge cursor appears. If you try to drop it on an inappropriate location, the cursor will display the ⦸ symbol. Both the dimension and the geometry return to their normal colors. The dimension's value updates to reflect the size of the geometry. If you need to change the dimension, double-click it.



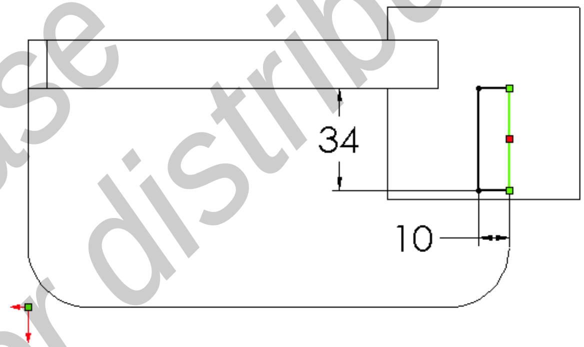
16 Exit the sketch to rebuild the model.

17 Remaining errors.
A few errors/warnings remain. The `Rib_Under` feature will be worked on next.

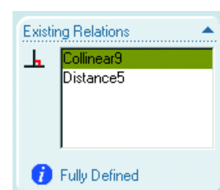
Edit the sketch of that feature.



18 Dangling relations.
One line of the sketch is shown in the dangling color. Click on that line to select it and display its drag handles. The red handle can be used in a drag and drop procedure, similar to what was done for the dangling dimension.



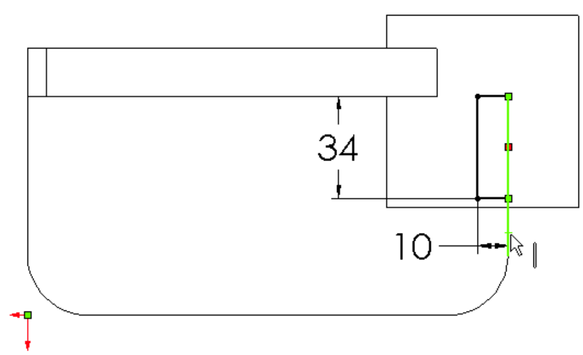
When you click on the line, its relations are displayed in the PropertyManager. The relation that is dangling is color coded the same as the sketch entity itself.



Note

If you double-click the dangling entity, callouts also appear in the graphics window.

19 Reattach.
Drag the red handle onto the rightmost vertical line of the `Base_Plate`. The system transfers the collinear relation from the missing entity (the deleted plane) to the model edge. The sketch is no longer dangling.



Repairing Relations Using Display/ Delete Relations

Some relations, like coincident points, can only be repaired through the **Display/Delete Relations** command. This option allows you to sort through all the relations in a sketch.

Introducing: Display/Delete Relations

Display/Delete Relations provides a way to systematically query all entities in a sketch. In addition, you can display the relations based on criteria such as dangling or over defined. You can also use **Display/ Delete Relations** to repair dangling relations.

Where to Find It

- Click **Tools, Relations, Display/Delete...**.
- Right-click in the sketch, and select **Display/Delete Relations**.
- Click **Display/Delete Relations** on the Sketch toolbar.

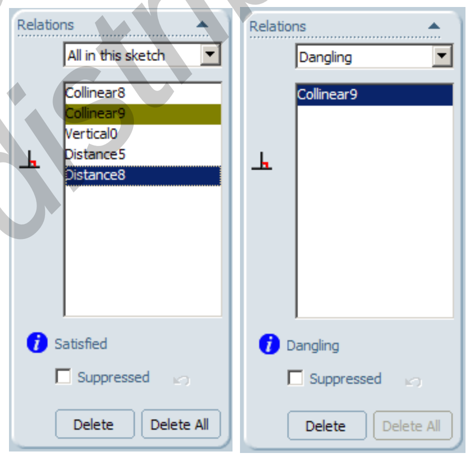
---

20 Undo.
Click **Undo** to remove the last event, the repair of the dangling relation.

21 Display/Delete Relations.
Right-click, and select **Display/ Delete Relations.**

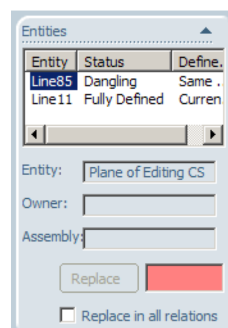
In the **Filter** list select **Dangling**. This displays only the relations that are dangling.

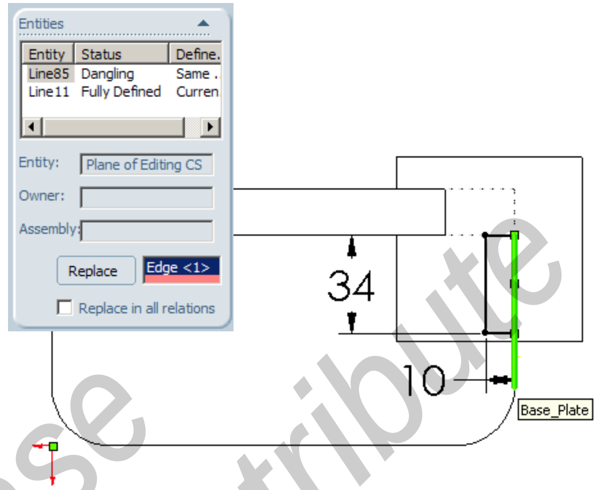
Select the **Collinear** relation.



22 Entities section.
Look at the lower section of the PropertyManager. There is a list of the entities used by this relation. One entity has a **Fully Defined** status, the other is **Dangling**.

Select the entity marked **Dangling**.





23 Replacement.
Select the vertical edge of the `Base_Plate`.

Tip

Select Other can be used to choose the edge.

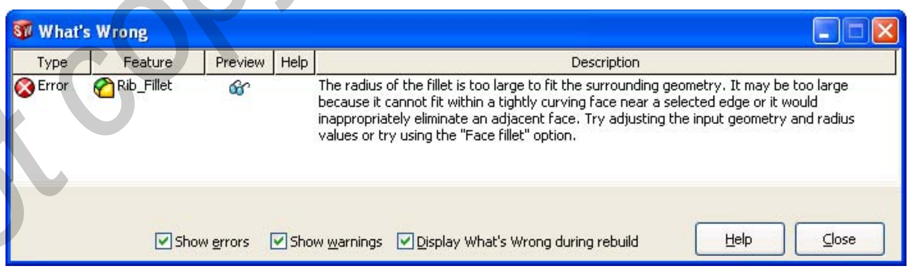
Click **Replace** and then click **OK**.

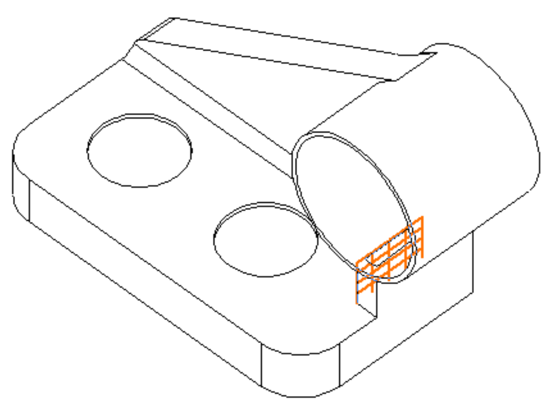
24 Exit the sketch.
Exit the sketch to see the one remaining error. The marker is placed on the feature `Rib_Fillet`. Use **What's Wrong** on the `Rib_Fillet` feature.

Highlighting Problem Areas

Certain error messages contain the preview symbol . If you click on that marker, the system will highlight the problem area in the model. If you use **What's Wrong** on the feature directly, it automatically highlights the problem area.

25 Highlight message.
Click on the preview symbol to visually display the area in which the error occurs.





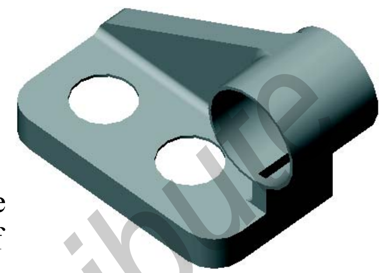
26 Graphic error display.
The area where the error occurs is highlighted with a mesh pattern. The fillet fails in this area. **Close** the message dialog.

27 Change the value.
Using the graphic and text information supplied by the system, it is clear that the problem lies in the radius value of the fillet. Set the value to something much smaller, for example **5mm**, and rebuild.

28 Model rebuilt.
The model is now rebuilt without any error or warnings.

Information From a Model

The part has some built-in problems related to the sequence of features. These problems will become evident when it comes time to make design changes. In order to understand the way that this part was constructed, we will walk through the steps of building it and introduce some of the tools that will be used. The design intent of the part will be revealed as the features are rebuilt one at a time.



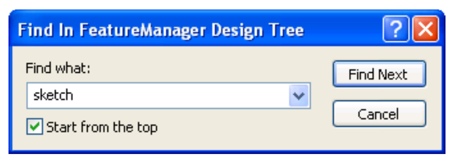
Introducing: Go To

The **Go To...** option can be used to search the text of the FeatureManager for a specific word or set of characters. Features are expanded to show any features found.

Where to Find It

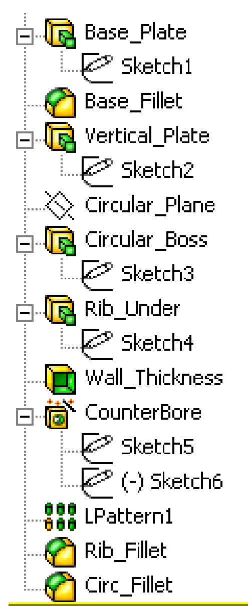
- Right-click the top level feature, and select **Go To...**.

29 Go To.
Right-click the top level feature and select **Go To...**. Type the partial name `sketch` and click **Start from the top**.



30 Find Next.
Click **Find Next** until the last occurrence is found. The message **This item was not found** will appear.

The search expanded all the features that have sketches so that the sketches are visible.



Tip
You can close all expanded features by right-clicking in the FeatureManager and choosing **Collapse Items**.

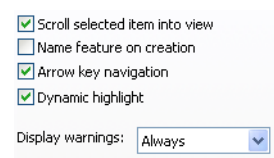
Introducing: The Rollback Bar

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.

Where to Find It

- Drag the rollback bar in the FeatureManager design tree.
- Or, right-click a feature, and select **Rollback** from the shortcut menu. This places the bar *before* the selected feature.
- Or, right-click in the FeatureManager and select **Roll To Previous** to move to the last position of the rollback bar. Select **Roll to End** to move the bar to after the last feature in the tree.
- Or, click **Tools, Options, System Options, FeatureManager** and click **Arrow key navigation**. This allows the arrow keys to move the rollback bar.

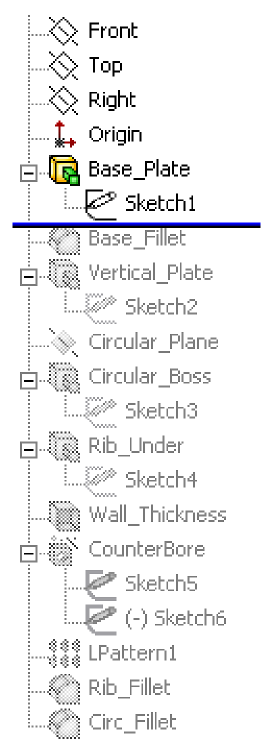
The focus must be set to the rollback bar by clicking on it. If the focus is set to the Graphics Area, the arrow keys will rotate the model.



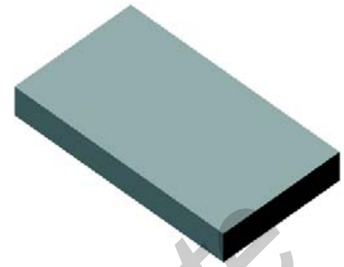
Note

The **Rollback** tool is also useful when editing large parts to limit rebuilding. Roll back to the position just after the feature that you are editing. When the editing is completed, the part is rebuilt only up to the rollback bar. This prevents the entire part from being rebuilt.

31 Roll the part back to the beginning.
Using **Rollback**, place the bar at the first feature in the FeatureManager design tree. This places the rollback bar after the feature `Base_Plate`. It can then be *rolled forward* one feature at a time.

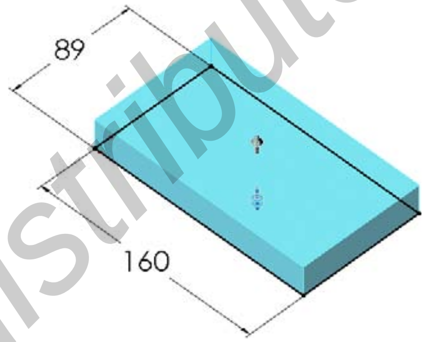


32 Feature Base_Plate.
The Base_Plate was created from a rectangle and extruded. To investigate this further, use **Edit Feature** on the feature.



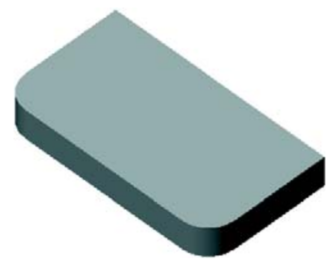
33 Edit Feature.
The graphics show the sketch geometry and the preview. **Cancel** the dialog.

Roll forward one feature by dragging the marker or moving it down with the arrow key.

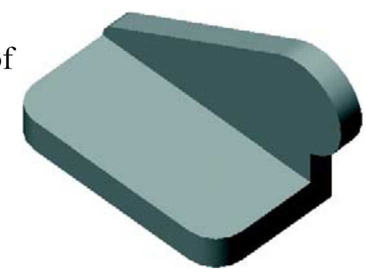


34 Feature Base_Fillet.
Fillets of equal radius are added to the front corners in this feature.

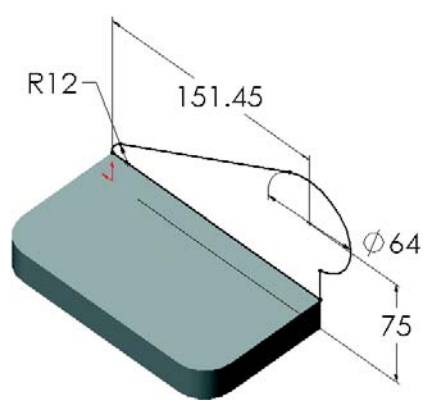
Roll forward to a position just before the Vertical_Plate feature.



35 Feature Vertical_Plate.
This feature was sketched on the rear face of the model and extruded towards the front.



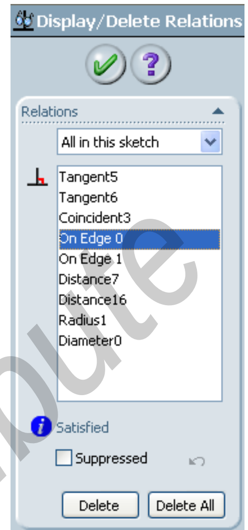
36 Edit Sketch.
Edit the sketch of the feature Vertical_Plate to see the geometry and its connections points.



37 Display/Delete Relations.

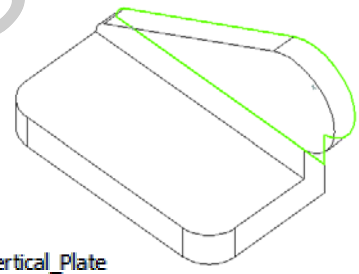
Click **Display/Delete Relations** . Set the **Filter** to **All in this sketch** and click individual relations in the list to explore all of geometric relations on the sketch entities. The relations will explain how entities are attached to each other and to the rest of the model.

Close the dialog and close the sketch without making any changes.



38 Sketch geometry.

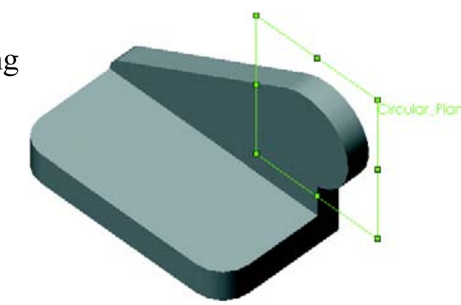
To see the sketch geometry more clearly, right-click `Sketch2`, and select **Show** sketch. The sketch icon appears in color when it is being shown. Using **Hidden Lines Removed**, the position of the sketch is clear.



Roll forward to a position just before the `Circular_Plane` feature.

39 `Circular_Plane`.

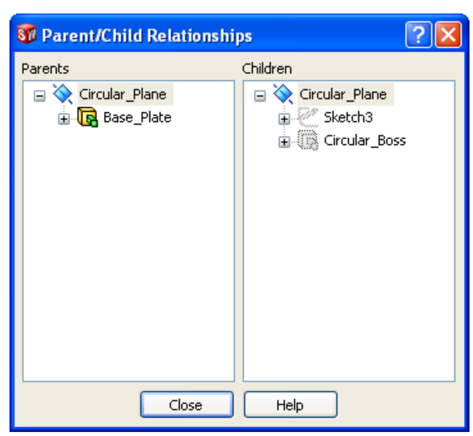
The plane was created for sketching the next feature, a circular boss. It lies behind `Sketch2`.



40 Parent/Child relationships.

Check the relationships on the plane. Right-click the plane and select **Parent/Child...**. The **Parent** of the plane is the `Base_Plate` feature – the plane is dependent upon it. The **Children** are `Sketch3` and the `Circular_Boss`; they are dependent on the plane.

Click **Close** and roll forward.



41 Feature `Circular_Boss`.
`Circular_Plane` was used for sketching `Circular_Boss`. The sketch was extruded through the part from the rear.

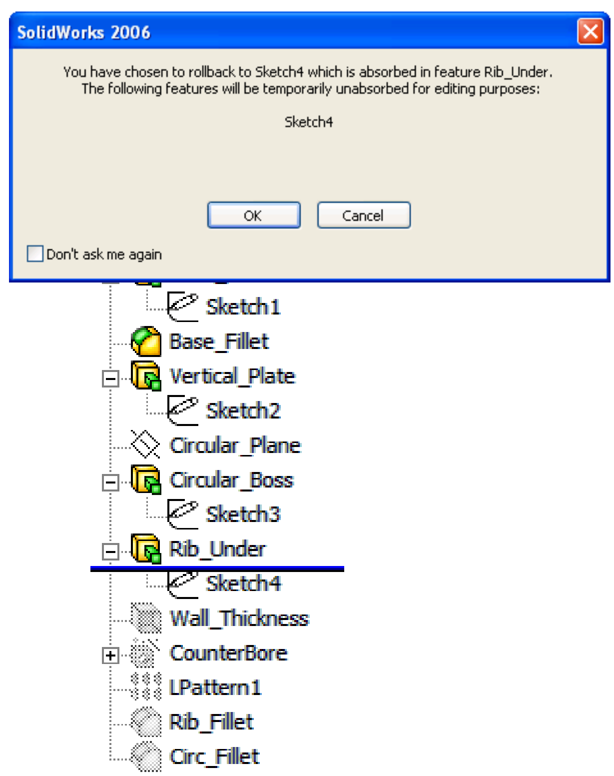
Roll forward to a position just before the `Wall_Thickness` feature.

42 Feature `Rib_Under`.
This feature was sketched as a rectangle and extruded up into the `Circular_Boss`.

Rollback to a Sketch

If the rollback bar is dragged and dropped between an absorbed sketch and its feature, a dialog appears. The dialog tells you that you have chosen to rollback to an absorbed feature and that the feature will be temporarily unabsorbed so it can be edited. This changes the sequence so that the sketch *precedes* the feature.

43 Rollback to `Sketch4`.
Move the rollback bar to a position between the `Rib_Under` feature and its sketch `Sketch4`. Click **OK** when the message appears.



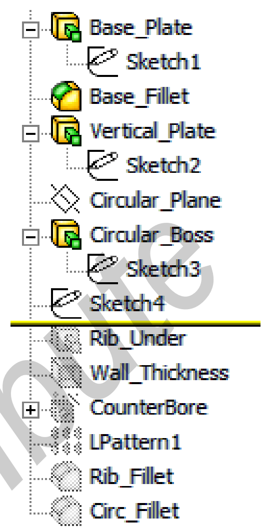
Tip

This technique is very useful when editing features that have multiple sketches such as **Sweep** and **Loft** features.

Sweeps and lofts are covered in the course *Advanced Part Modeling*.

44 Roll Forward.

Roll forward to a position after `Sketch4`.



Introducing: Edit Sketch Plane

Edit Sketch Plane allows you to change the plane or face that a specific sketch is created on. The new sketch plane does not have to be parallel to the original.

Where to Find It

- From the **Edit** menu, select **Sketch Plane...**.
- Or, right-click the sketch select **Edit Sketch Plane...**.

---

45 Edit Sketch Plane.

Right-click the `Sketch4` feature and choose the option **Edit Sketch Plane** to determine which sketch plane was used. The highlighted face identifies the sketch plane.

Click **Cancel** and roll forward to a position after the `Wall_Thickness` feature.

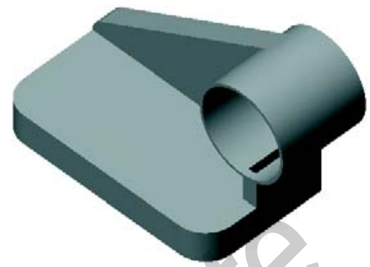
Note

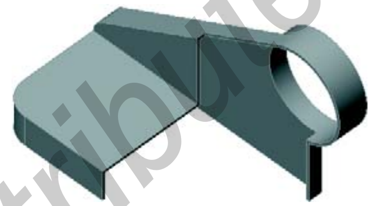
The selection for **Sketch Plane/Face** can be cleared to force the sketch plane reference to dangle. A confirmation message appears in that case.

`There is no selection for the sketch plane reference. Select OK if you would like this to be a gangling reference.`

46 Feature `Wall_Thickness`.
The model was shelled out leaving both circular faces and the bottom face open. See the section cut at the right for details.

Roll forward to a position after the `CounterBore` feature.



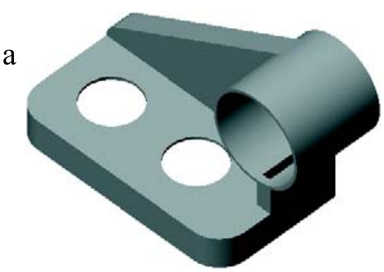


47 Feature `CounterBore`.
The **Hole Wizard** was used to create a counterbore hole on the top planar face. However, due to the thin wall, it appears as a simple cut.

Roll forward to a position after the `LPattern1` feature.

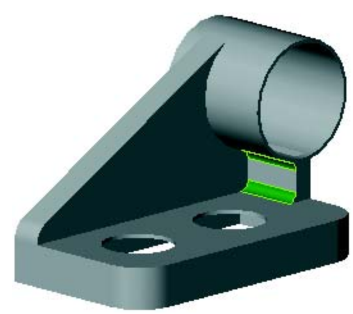
48 Pattern feature.
The `CounterBore` was patterned using a linear pattern, `LPattern1`.

Roll forward to a position after the `Rib_Fillet` feature.

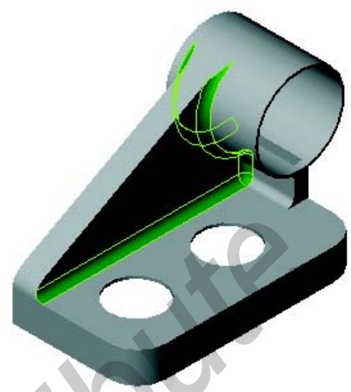


49 `Rib_Fillet` feature.
The `Rib_Fillet` feature creates large fillets where the `Rib_Under` joins the `Circular_Boss` and `Base_Plate`.

Right-click and select **Roll to End**.



50 **`Circ_Fillet` feature.**
This feature creates smaller fillets on both sides of the `Vertical_Plate`.

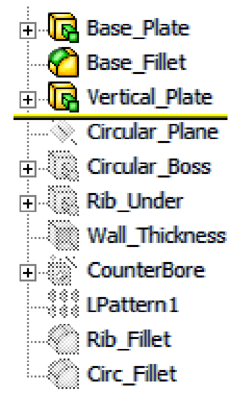


Rebuilding Tools

Rebuilding a model incorporates the changes that you have made. Slow rebuild times can slow down the modeling process significantly. There are some tools available to optimize rebuilding times.

Rollback to Feature

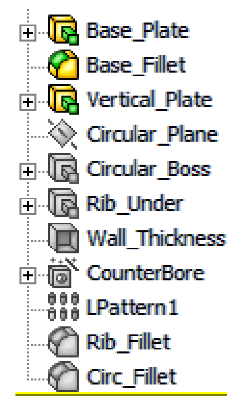
Rollback can be used to limit the rebuilding time by rolling back to the feature being edited. For example, if the `Vertical_Plate` is being edited, rollback to a position just after that feature.



Changes are made to the feature and it is rebuilt. Due to the rollback position, only the features *before* the bar are rebuilt, limiting the scope of the rebuild. The remainder of the part will be rebuilt when the bar is moved or when the part is saved.

Feature Suppression

Feature Suppression is a more permanent method of limiting rebuild time. Features that are suppressed are not rebuilt. Configurations can be used to arrange combinations of suppressed features.



Rebuild Feedback and Interrupt

During a rebuild, a progress bar and status are shown on the bottom bar of the SolidWorks window. The rebuild can be stopped by pressing the **Esc** (Escape) key.

Feature Statistics

Feature Statistics is a tool that displays the amount of time it takes to rebuild each feature in a part. Use this tool to identify the features that take a long time to rebuild. Once they are identified, you can possibly edited them to increase efficiency, or suppress them if they are not critical to the editing process.

Introducing: Feature Statistics

The **Feature Statistics** dialog box displays a list of all features and their rebuild times in descending order.

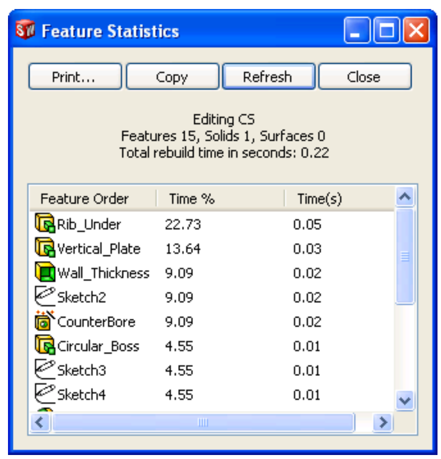
- **Feature Order**
  Lists each item in the FeatureManager design tree: features, sketches, and derived planes. Use the shortcut menu to **Edit Feature**, **Suppress** features, and so on.
- **Time%**
  Displays the percentage of the total part rebuild time to regenerate each item.
- **Time**
  Displays the amount of time in seconds that each item takes to rebuild.

Where to Find It

- From the menu, select **Tools, Feature Statistics...**.

51 Feature Statistics.
Click **Tools, Feature Statistics...**.

The features are listed in descending order according to the amount of time required to regenerate them.



Interpreting the Data

The first thing to keep in mind is that the total rebuild time for this part is approximately $^1/_3$ second, so a change to any one feature is not likely to make a significant difference.

The second thing is the number of significant digits and rounding error. For example, `Feature1` may appear to take twice as long to rebuild as `Feature2`, 0.02 seconds versus 0.01 seconds. Does this indicate a problem with `Feature1`? Not necessarily. It is quite possible that `Feature1` takes 0.0151 seconds while `Feature2` takes 0.0149 seconds, a difference of only 0.0002 seconds.

Use **Feature Statistics** to identify features that significantly impact rebuild time. Then either:

- Suppress features to improve performance.
- Analyze and modify features to improve performance.

What Affects Rebuild Time?

Features can be analyzed to determine why they behave as they do. Depending on the feature type and how it used, the reasons will vary.

For sketched features, look for external relations and end conditions that reference other features. Keep these relations attached to the earliest feature possible. Do the same for sketch planes.

Tip

In general, the more parents that a feature has, the slower it will rebuild.

See *Repairing Relations Using Display/Delete Relations* on page 231 for an example of changing relations in a sketch.

For features applied to edges or faces, check the feature's options and the position of the feature in the FeatureManager. See *Edit Feature* on page 253 for an example of changing relations in a feature.

In general, there are four tools available to modify features:

- **Edit Feature**
- **Edit Sketch**
- **Edit Sketch Plane**
- **Delete Feature**

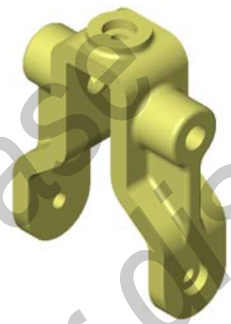
Exercise 27: Errors1

Edit this part using the information and dimensions provided to repair the errors and warnings and complete the part.

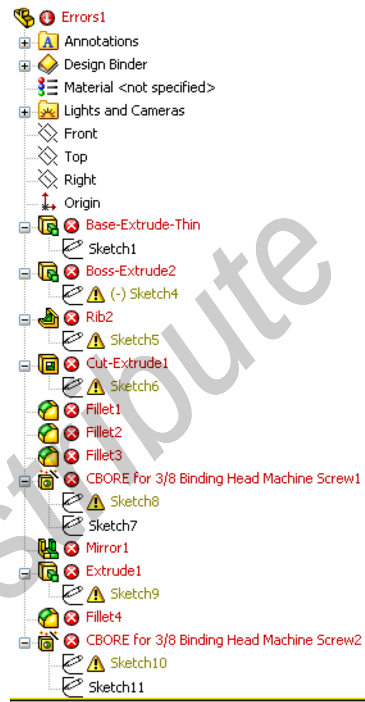
This lab reinforces the following skills:

- Using **What's Wrong?**
- **Edit Sketch**.
- Adding geometric relations.
- **Edit Feature**.
- Fixing rebuild errors.

Before

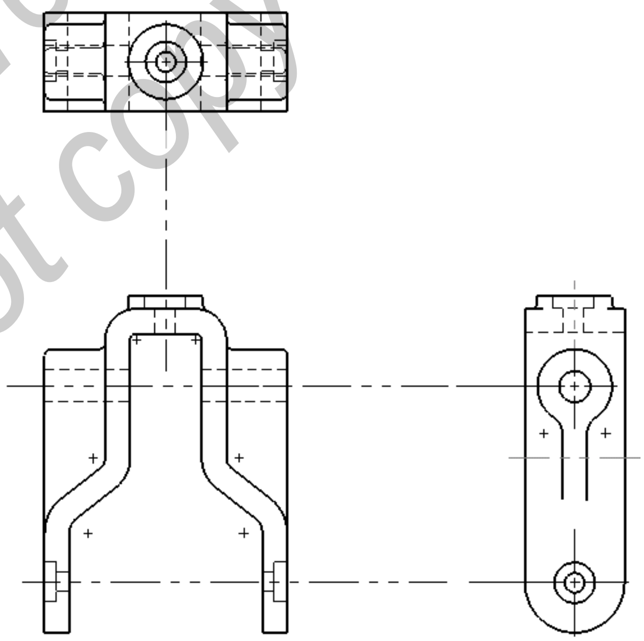


After



Procedure

Open the existing part `Errors1` and make several edits to remove the errors and warnings from the part. Use the drawing below as a guide.



Tip

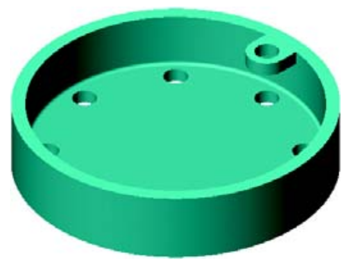
Use **Merge solids** in the `Mirror1` feature. The completed part should be a *single* solid body.

Exercise 28: Errors2

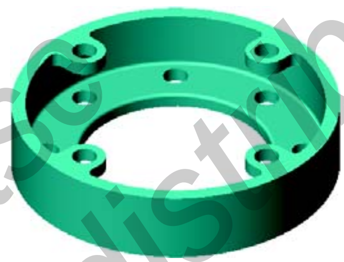
Edit this part using the information and dimensions provided to repair the errors and warnings and complete the part.

This lab reinforces the following skills:

- Using **What's Wrong?**
- **Edit Sketch**.
- Adding and deleting geometric relations.
- **Edit Feature**.
- Fixing rebuild errors.



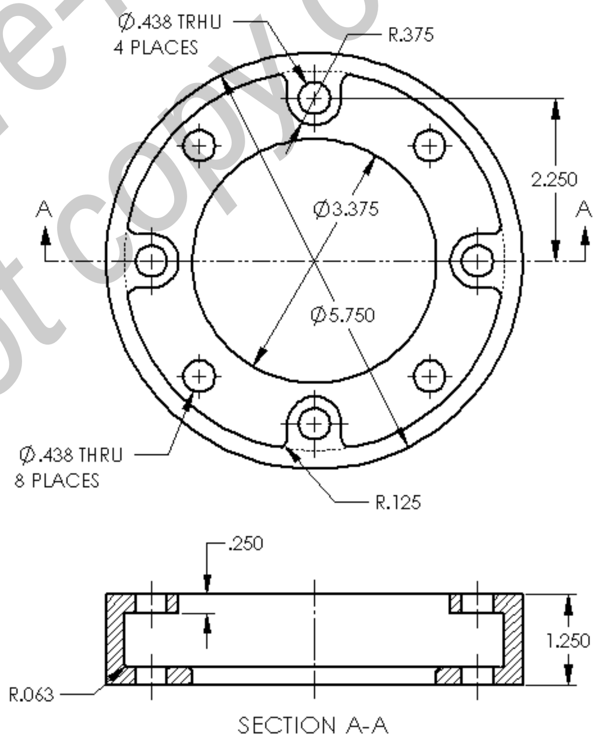
Before



After

Procedure

Open the existing part `Errors2` and make several edits to remove the errors and warnings from the part. Use the drawing below as a guide.



Exercise 29: Copy and Dangling Relations

Complete this part by copying features and making repairs.

This lab uses the following skills:

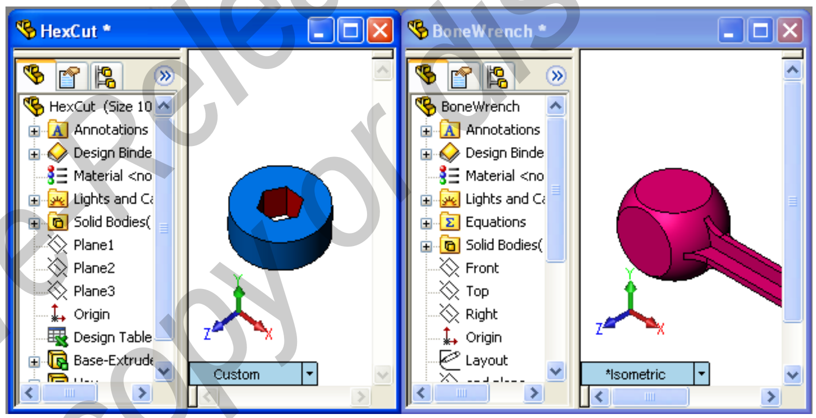
- Copying features between parts.
- Editing sketches to repair dangling relations.

Units: **millimeters**

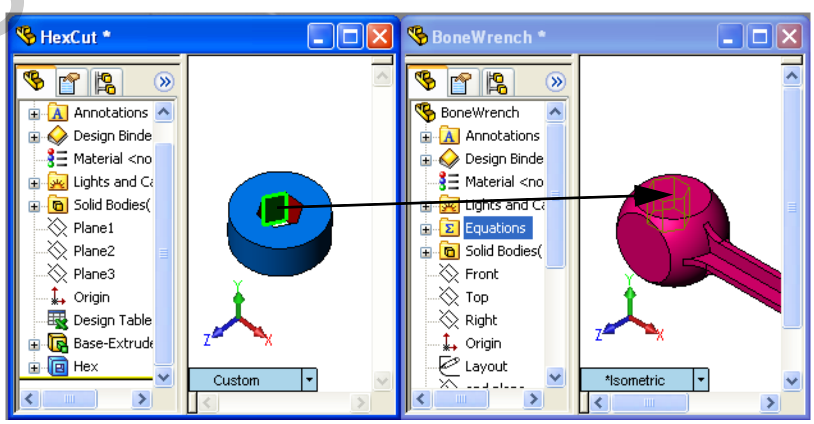
Procedure

Open existing parts.

1. **Open the `BoneWrench` and `HexCut`.**
   Open both files and click **Window, Tile Vertically**.



2. **Copy `Hex` feature.**
   Control-Drag a face of the `Hex` feature and drop it onto the top planar face of the `BoneWrench` as shown.

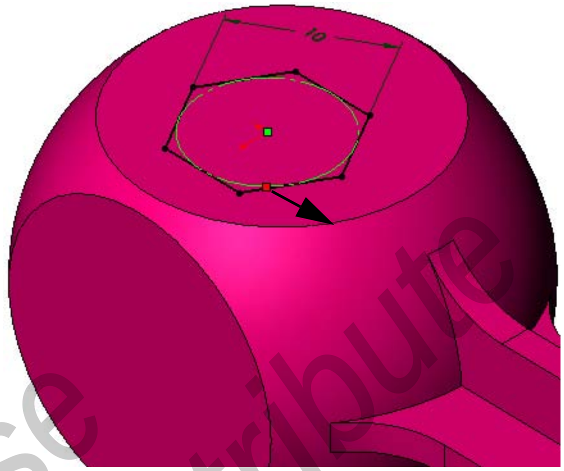


Click the **Dangle** button on the **Copy Confirmation** dialog.

3 Repair.
Edit the sketch with an error and select the inner construction circle.

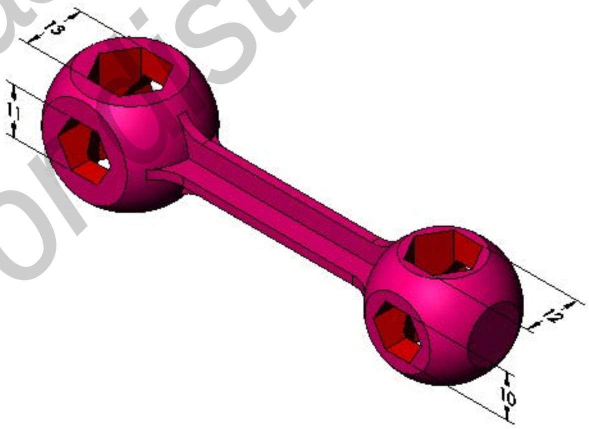
Drag and drop the red marker to an appropriate replacement reference.

Size the hexagon as shown below.

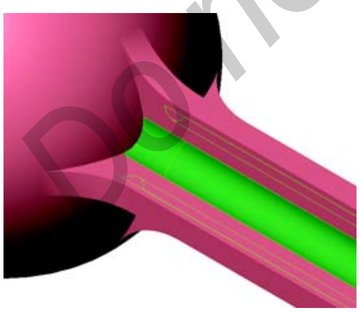
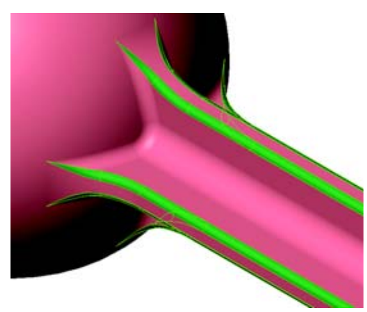


Locations

Use the following graphics to locate and size a total of 4 copied cuts using the end condition **Through All**.



4 (Optional)Cosmetic fillets.
Optionally add the following fillets and rounds:

R2mm	R1mm	R0.5mm
		

5 Save the parts and close them.

Lesson 8
Editing: Design Changes

Upon successful completion of this lesson, you will be able to:

- Understand how modeling techniques influence the ability to modify a part.
- Use Sketch Contours to define the shape of a feature.

Part Editing

The SolidWorks software provides the capability to edit virtually anything at any time. In order to emphasize this, the major tools for editing parts are used here to create a design change.

Stages in the Process

Some key stages in the process of modifying this part are shown in the following list. Each of these topics comprises a section in the lesson.

- **Delete and rename feature**
  The Delete key is one of the most straightforward editing tools. Renaming features is useful for later use of the part.
- **Use editing tools**
  Use common editing tools such as Edit Feature, Reorder and Edit Sketch to modify the geometry and design intent.
- **Sketch contours**
  A single sketch can be used to create multiple features by using contours within the sketch.
- **Adding textures**
  Texture maps can be added to the entire part or selected faces to provide a realistic appearance of materials or threads.

Design Changes

Some changes have to be made to the model. Some will change the structure of it, others only dimension values. Making design changes to a model can be as simple as changing the value of a dimension and as difficult as removing external references. This section steps through a series of changes to a model. The focus is on editing features rather than deleting and reinserting them. Editing allows you to maintain references to drawings, assemblies or other parts that would be lost if you deleted the feature.

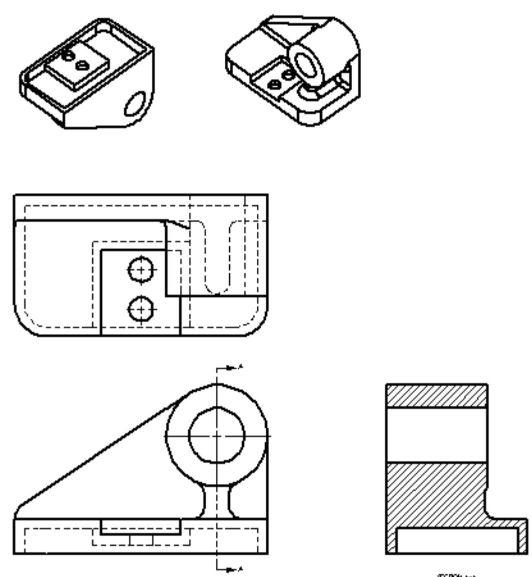
Procedure

We will continue editing the part that was repaired in the previous lesson.

Required Changes

The changes to the model are as follows:

- The circular boss is centered over the rib.
- The rib is rounded at the end.
- The circular boss is tangent to the right edge.
- A cutout with holes is added to the base.
- Both holes are equal radius.
- Only the base is shelled.



Deletions

Any feature can be deleted from the model. Consideration should be given to what other features, other than the selected one, will be deleted with it. The **Confirm Delete** dialog lists **Dependent Items** that will be deleted with the selected one. The sketches of most features are not automatically deleted. However, the sketches associated with **Hole Wizard** features *are* automatically deleted when the hole is deleted. For other dependent features, deleting the parent will delete the children.

1 **Open the part `Editing CS Repaired`.**

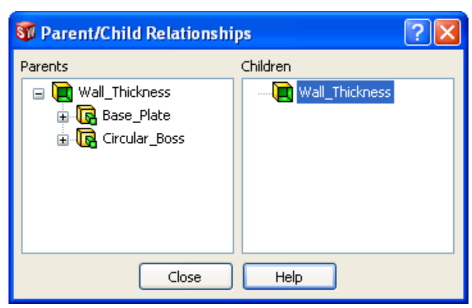
2 **Delete feature.**
Select and delete the `CounterBore` feature. The check box, **Also delete all child features**, is already checked. The dialog indicates the `LPattern1` feature will also be deleted because it is a child of the `CounterBore`.

Click **Yes** to confirm the deletion.

3 **Try to reorder.**
Try to reorder the shell feature, `Wall_Thickness`, to a position immediately after the `Base_Fillet`. The cursor displays a "no move" symbol and a dialog appears telling you that you cannot reorder because of parent/child relations. You cannot reorder a child before the parent.

Click **OK**.

4 **Parent/Child.**
Select the `Wall_Thickness` feature and click **Parent/Child...** from the right mouse menu. The dialog shows that the parents of the `Wall_Thickness` feature are `Base_Plate` and `Circular_Boss`.

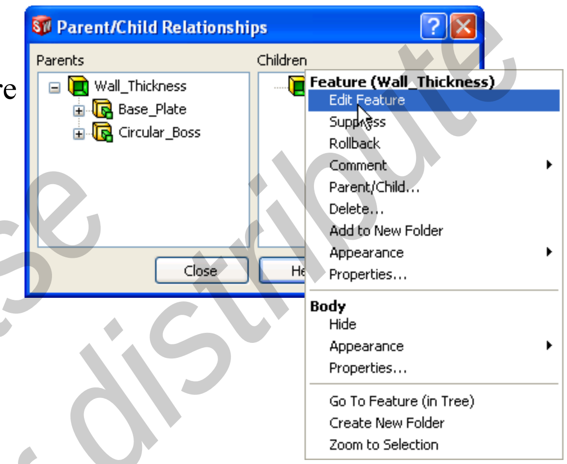


The `Circular_Boss` references need to be removed in order for us to be able to reorder the feature.

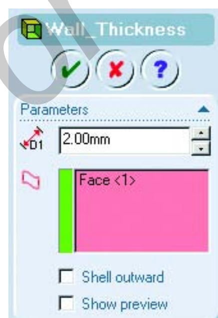
Edit Feature

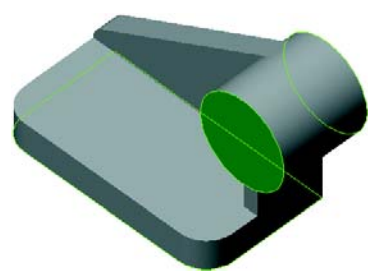
Edit Feature allows you to change a feature using the same dialog and user interface that was used to create it. Simple changes, like dimension values or directions, can be made along with more complex ones such as the removal or addition of selections.

5 Edit Feature.
Right-click the `Wall_Thickness` feature from the **Parent/Child...** dialog and select **Edit Feature**.



Select both of the highlighted circular faces. The **Faces to Remove** selection list will show only a single face.





When you reselect an already selected face, it acts like a toggle, deselecting it.

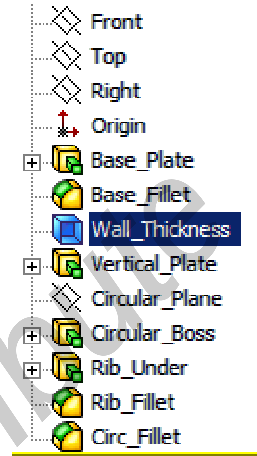
As an alternative, you can click on an item in the selection list and deselect it by pressing the **Delete** key on the keyboard. Sometimes this can be confusing because you might not always know which face is labeled `Face<2>`.

6 Changes to Parent/Child.
Editing the `Wall_Thickness` feature causes a change in the **Parent/Child Relationship**. The **Parents** section now lists only one feature, the `Base_Plate`.

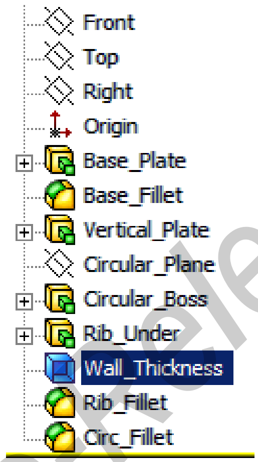
Reorder

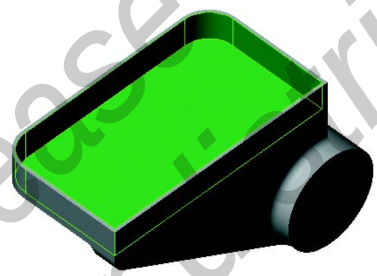
Reorder allows for changes to the sequence of features in the model. Sequence changes are limited by parent/child relationships that exist.

7 Reorder.
Drag the `Wall_Thickness` feature and reorder it by dropping it on the `Base_Fillet`. It is positioned after the `Base_Fillet` feature.



8 Results.
Now the shelling operation affects only the first and second features of the part.

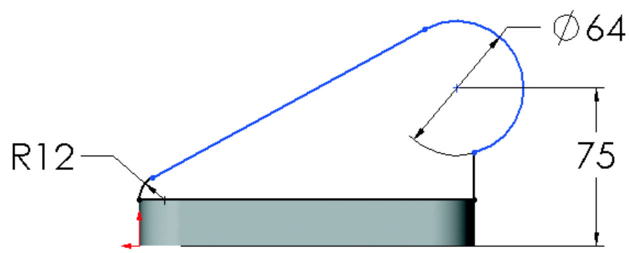




Edit Sketch

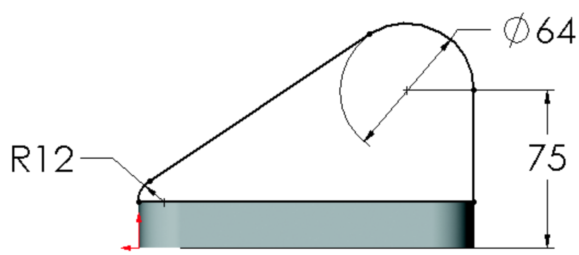
Edit Sketch opens the feature's sketch for changes to dimension values, dimensions, and relations. In addition, geometry can be removed or added to the sketch.

9 Editing the sketch.
Edit the sketch of the `Vertical_Plate` feature.



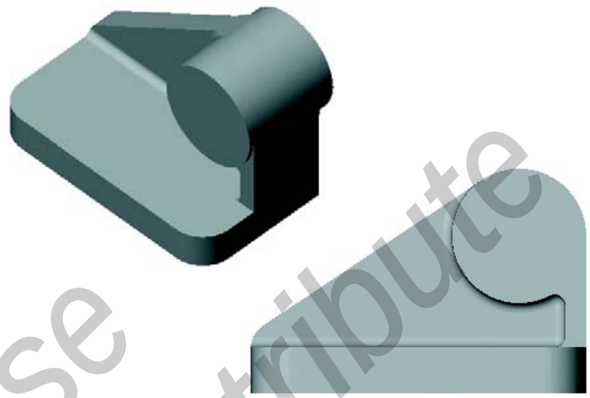
10 Delete dimension.
Delete the horizontal linear dimension. This will cause the sketch to become under defined.

11 Add new relation.
Hold down **Ctrl** and select the rightmost vertical line and the arc. Right-click and select **Tangent**. This adds a **Tangent** relation between the line and the arc.

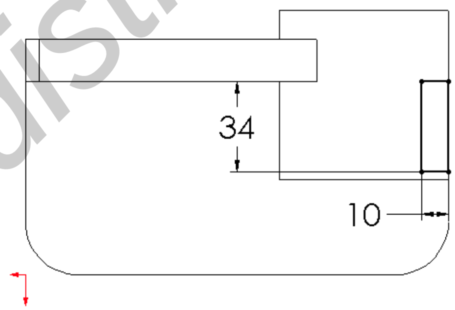


12 Exit the sketch.

13 Resulting model.
This moves the `Circular_Boss` so that its cylinder face is tangent to the outer edge of the `Base_Plate`. The fillets update to the new positions.

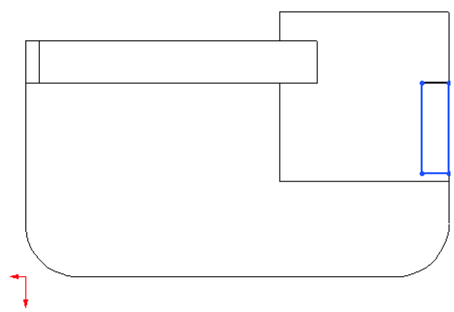


14 Edit the `Rib_Under` sketch
The `Rib_Under` sketch is still tied to its original relations, the outer edge of the `Base_Plate`.



15 Edit the sketch.

16 Display relations.
Show all the geometric relations in the sketch using the **All in this sketch** option. In order to reposition the rib, most of the relations must be deleted.

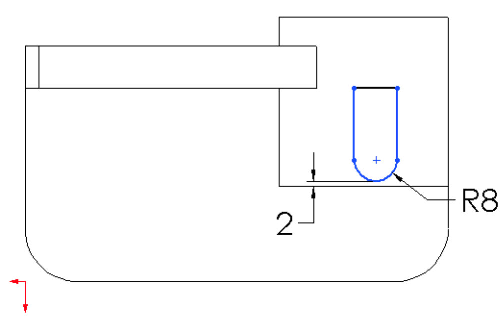


Select and remove these relations using the **Delete** button:

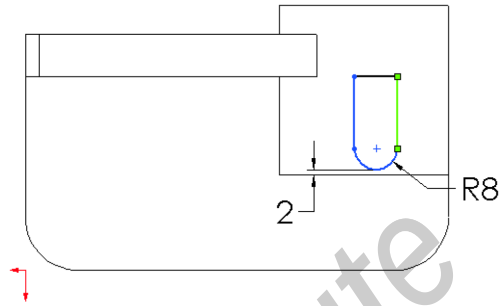
- **Collinear** relation to the *vertical* edge of the `Base_Plate`.
- Both **Distance** relations (the two dimensions).

Keep the **Collinear** relation to the `Vertical_Plate` and the **Vertical** relation on the left hand line.

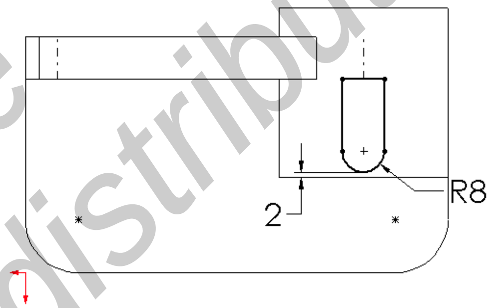
17 New geometry.
Delete the bottom line of the rectangle and add a tangent arc. Dimension the sketch as shown.



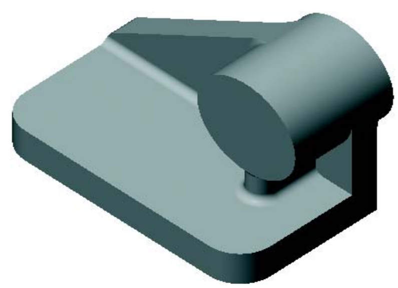
18 Vertical relation.
Deleting the **Collinear** relation leaves the right vertical line without any relation to keep it vertical. To fix this, add a **Vertical** relation to the rightmost line.



19 Temporary graphics.
Turn on display of **Temporary Axes** and relate the center of the arc to the temporary axis. This will center the rib on the circular boss.

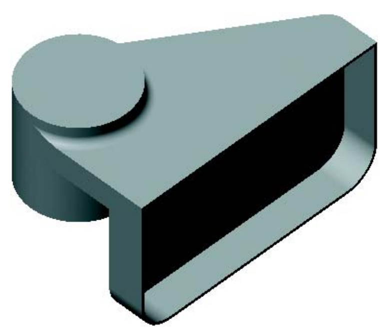


20 Result.
The `Rib_Under` feature is now centered under the `Circular_Boss`. It has a rounded front edge and is also inside the edge of the boss by a small amount.



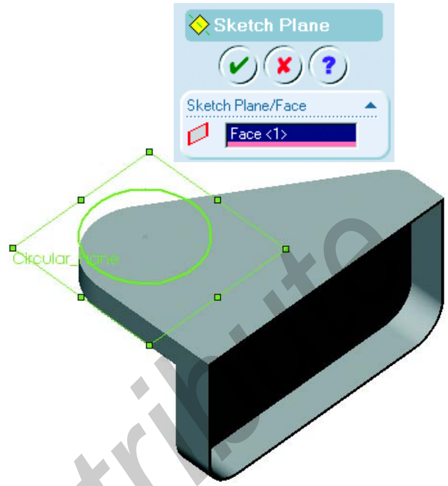
21 Edit Sketch Plane.
Expand the listing of the `Circular_Boss` feature. Right-click the sketch and select **Edit Sketch Plane** from the shortcut menu.

You do not have to edit the sketch.

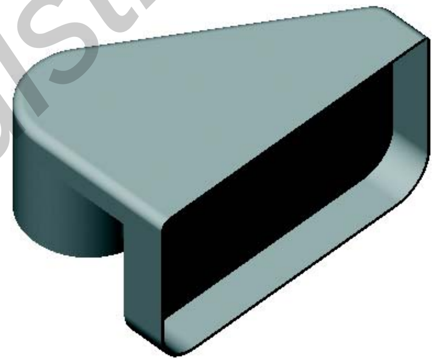


22 Face or plane selection.
The current plane used in the sketch is highlighted along with the sketch geometry. You can now choose a new sketch plane.

Select the rear face of the model and click **OK**.



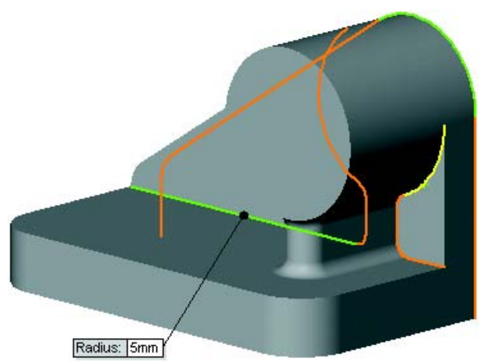
23 Edited sketch plane.
The `Circular_Boss` feature has been edited. The sketch now references a model face rather than a plane.



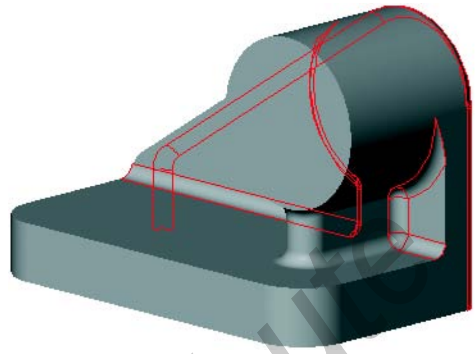
24 Delete the plane.
Check the **Parent/Child Relationships** of the plane. The `Circular_Plane` now has no children.

Delete the plane.

25 Edit Feature.
Edit the `Circ_Fillet` feature. Add the edge shown and click **Apply**.



26 Result.
The additional edge is filleted as part of the `Circ_Fillet` feature.



27 Save and Close.
An existing part will be used for the reminder of the case study.

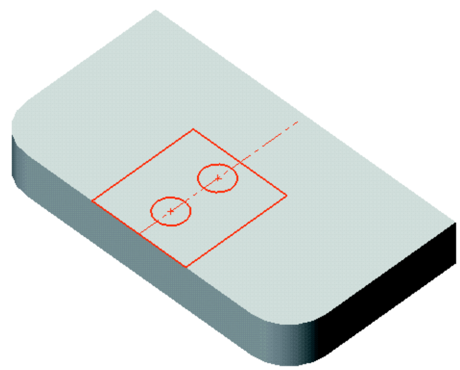
Rollback

Rollback is a tool that has many uses. Previously, it was used to "walk through" a model showing the steps that were followed to build it. It is also useful to add features at a specific point in the part's history.

28 Open `Partial_Editing` CS.
Open an existing part that is identical except for one additional sketch, `Contour Selection`. The sketch contains two circles enclosed within a rectangle.

29 Reorder and rollback.
Reorder the `Contour Selection` sketch to a position between the `Base Fillet` and `Wall_Thickness` features.

Rollback to a position between the `Contour Selection` sketch and `Wall_Thickness` feature.



Sketch Contours

Sketch Contours allow you to select portions of a sketch that are generated by the intersection of geometry and create features. This way you can use a partial sketch to create features.

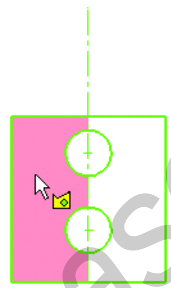
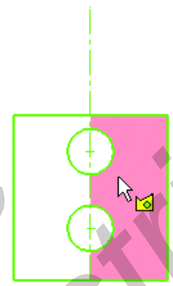
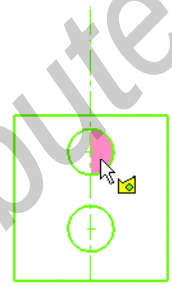
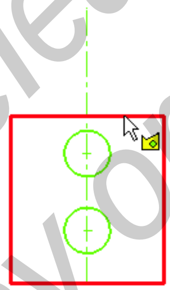
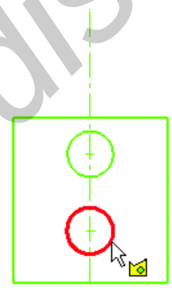
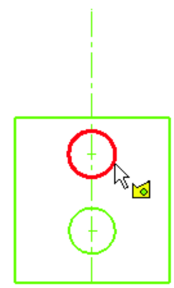
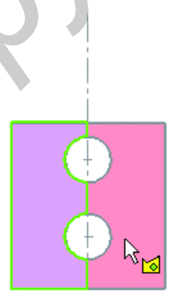
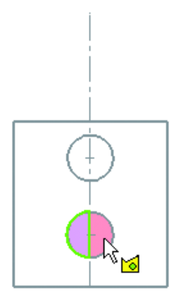
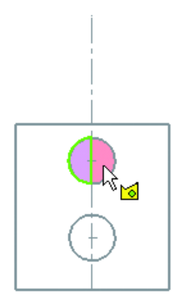
Another advantage of this method is that the sketch can be reused, creating separate features from different portions of the sketch.

Two commands, **Contour Select Tool** and **End Select Contours**, are used to start and end the contour selection process.

Contours Available

There are often multiple **Sketch Contours** available within a single sketch. Any boundary generated by the intersection of sketch geometry can be used singly or in combination with other contours.

Using this sketch as an example, there are some of the possible regions, contours and combinations available for use.

Individual Regions			
Individual Contours			
Combined Contour Selection			

Introducing: Contour Select Tool

The **Contour Select Tool** is used to select one or more contours for use in a feature. The cursor looks like this:  when the **Contour Select Tool** is active.

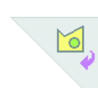
Where to Find It

- Right-click in the graphics area and choose **Contour Select Tool**.
- Right-click a sketch and choose **Contour Select Tool**.

Introducing: End Select Contours

End Select Contours is used end the selection of contours.

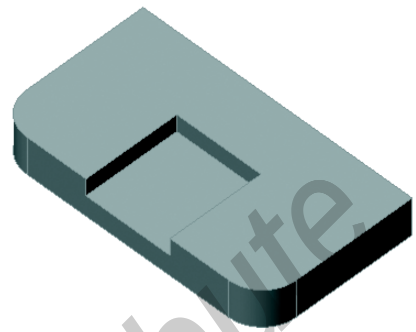
Where to Find It

- Right-click in the graphics area or on the sketch in the FeatureManager design tree and choose **End Select Contours**.
- Click the confirmation corner symbol .

30 Extrude a cut.
Use the **Contour Select Tool** to select the rectangular region of the sketch.

Create a blind cut, **10mm** deep into the model.

Rename the feature `Hole_Mtg`.



Shared Sketches

A sketch can be used more than once to create multiple features.

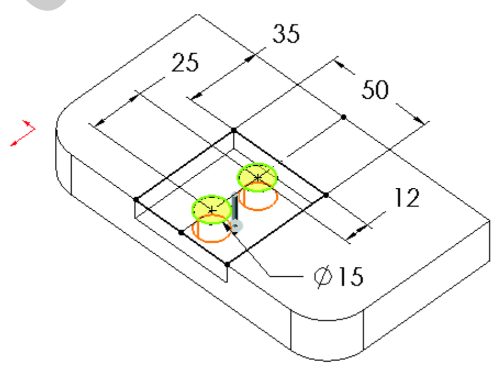
When you create a feature, the sketch is absorbed into the feature and hidden from view. When you activate the **Contour Select Tool**, the sketch is automatically made visible.

31 Add more cuts.
Select the sketch of the `Hole_Mtg` feature and click **Extruded Cut** on the Features toolbar.

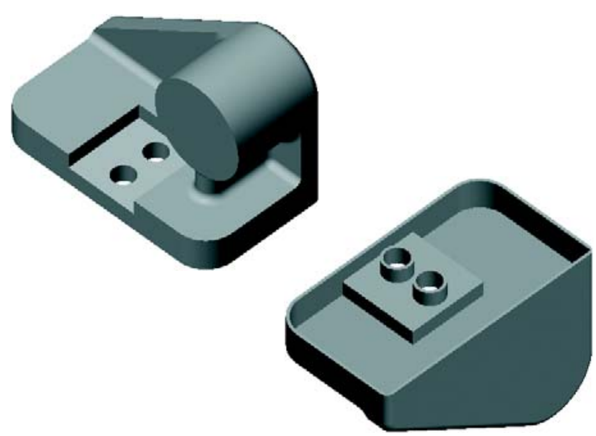
Expand the **Selected Contours** list and select the two circular regions of the sketch.

Extrude the regions using the end condition **Through All.**

Rename the cuts `Thru_Holes`.

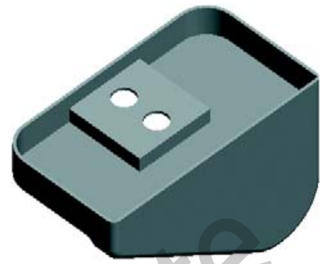


32 Roll to End.
Right-click in the FeatureManager design tree and select **Roll to End**. Note that the cut holes are used in the shelling operation to create additional, unneeded faces.



33 Reorder.

Reorder the `Thru_Holes` feature to a position after the `Wall_Thickness` feature. The result is that the `Thru_Holes` feature is not affected by the shelling.



34 Change wall thickness.

Change the wall thickness to **6mm** and rebuild to complete the model.

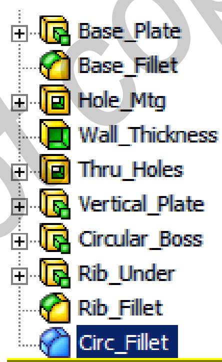
Copying Fillets

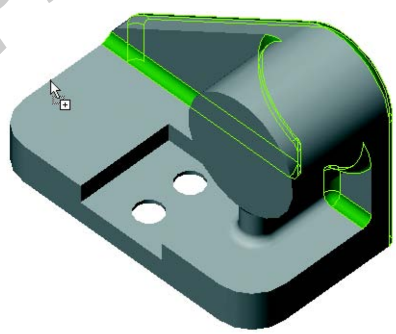
A quick and easy way to create a new fillet is to copy it from an existing feature. The new fillet is the same type and size but unrelated to the original.

35 Copy.

Hold down **Ctrl** and drag the `Circ_Fillet` feature onto the edge of the model. Release the mouse button.

The fillet can be copied from the FeatureManager design tree, or directly from the model.



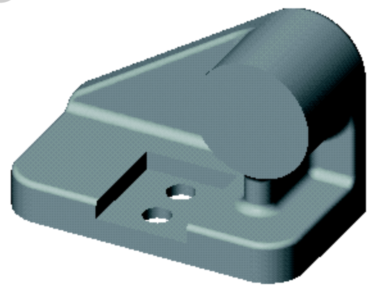


36 New fillet feature.
A new fillet feature is created on the edge.

Edit the fillet and add the edge on the opposite side. Change the radius value to **3mm**.

Chamfers can be copied using the same procedure.

37 Trimetric view.
Change the view orientation to a `Trimetric` view.



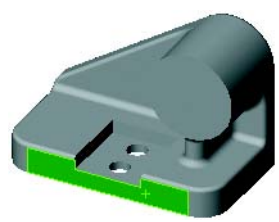
Introducing: Section View

Section View cuts the view using one or more section planes. The planes can be dragged dynamically. Reference planes or planar faces can be used.

Where to Find It

- Click **Section View** on the View toolbar.
- Or, click **View, Display, Section View**.

38 Select Face.
Select the planar face indicated. It will be used to define the section plane.



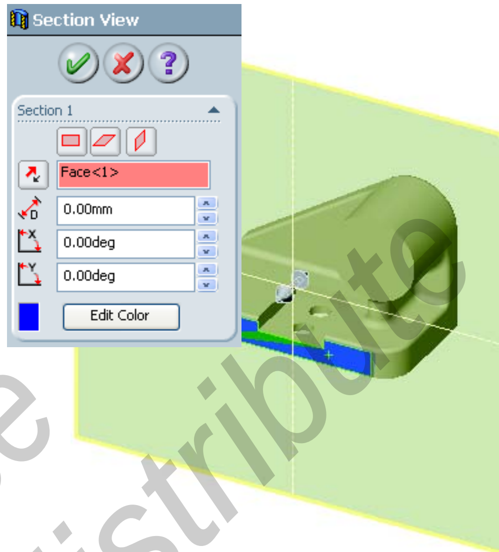
Note

You do not have to pre-select the section plane. If you do not, the system will use a default section plane, usually the `Front`.

39 Section view.

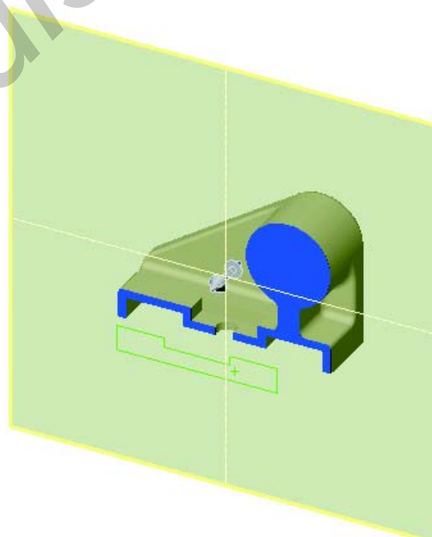
Click **Section View** to use the selected face as the section plane.

The **Section 1** group box includes options for the **Reference Section Plane**, **Section Direction**, **Offset Distance** and **Angles**.



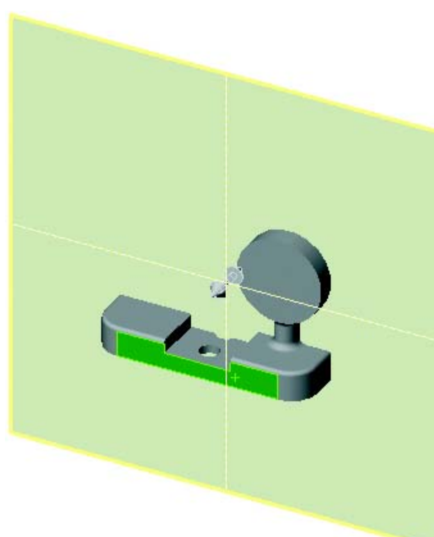
40 Drag the plane.
Using the arrows, drag in a direction normal to the plane and drop.

The plane angle can be changed by dragging the edges of the section plane.



41 Reverse section direction.
Click **Reverse Section Direction** to reverse the direction of the section.

Click **Cancel** to close the dialog.



Adding Textures

Make the model look more realistic by adding **Textures** to faces, surfaces, bodies, features and components. The quality of the texture is influenced by the graphics adaptor. For more information, see *RealView Graphics* on page 193.

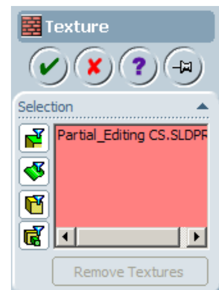
Where to Find It

- Click **Edit Texture** on the Standard toolbar.
- Or, right-click a face, feature or part and choose **Appearance, Texture...**.

Note

The textures are divided into folders such as **Metal, Plastic** and **Stone** with sub-folders.

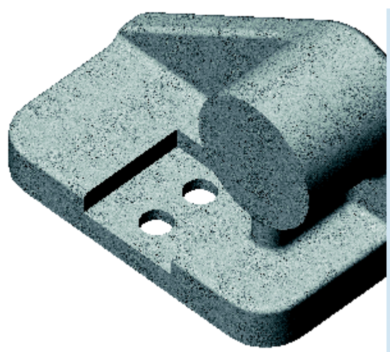
42 Geometry selection.
Right-click the top level feature (part name) and choose **Appearance, Texture...**.

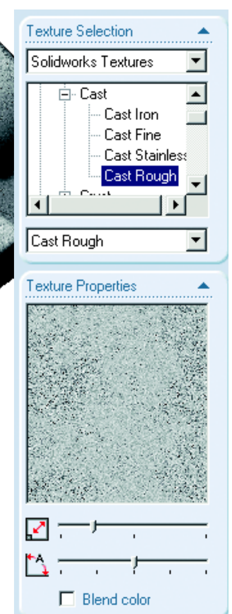


43 Texture selection.
Click the folders **Metal** and **Cast**. Select the texture **Cast Rough** from the **Texture Selection** list.

The texture is previewed in the **Texture Properties** group box.

All faces of the selected solid body share the same texture.

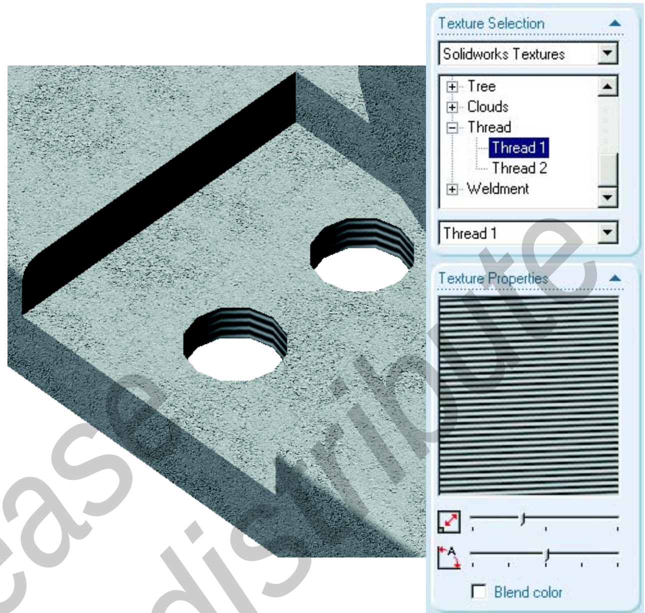




44 Thread texture.
Select the two "hole" faces and right-click **Appearance, Texture...**.

Choose the folder **Thread** and the texture **Thread1**.

Adjust the **Scale** of the texture using the **Scale** slider.

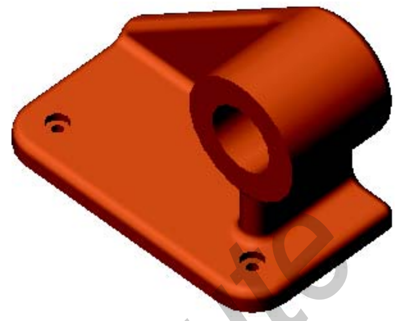


Exercise 30: Changes

Make changes to the part created in the previous lesson.

This exercise uses the following skills:

- Deleting features.
- Using **Link Values**.
- Reordering features.

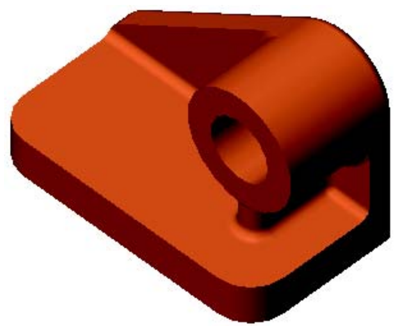


Procedure

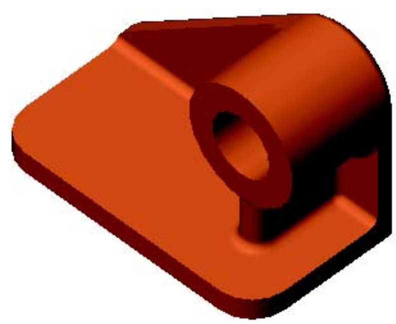
Use the following procedure:

1 **Open the part Changes.**
Several changes and additions will be made to the model.

2 **Delete.**
Delete the mounting holes, cutout and shell (Cut-Extrude1, Wall_Thickness and Cut-Extrude2) from the model.

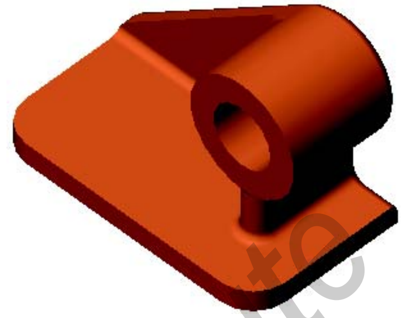


3 **Equal thickness.**
Set the thicknesses of the Base_Plate and Vertical_Plate to the same value, **12mm**, using **Link Values**.

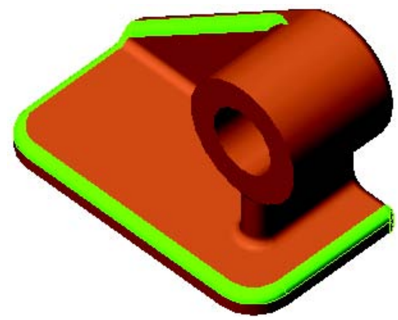


4 Cut.
Remove the portion of the `Vert_Plate` on the right side of the `Circular_Boss` and `Rib_Under`.

Edit and **Reorder** features where necessary to maintain the filleting.



5 Fillet.
Add another fillet the same radius as the `Circ_Fillet`.



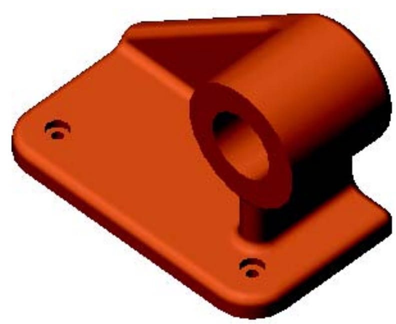
6 Counterbored holes.
Add two counterbored holes of the following size:

ANSI Metric

M6 Hex Cap Screw

Through All

Reorder features where necessary to avoid undercuts.



7 Save and close the part.

Exercise 31: Adding Draft

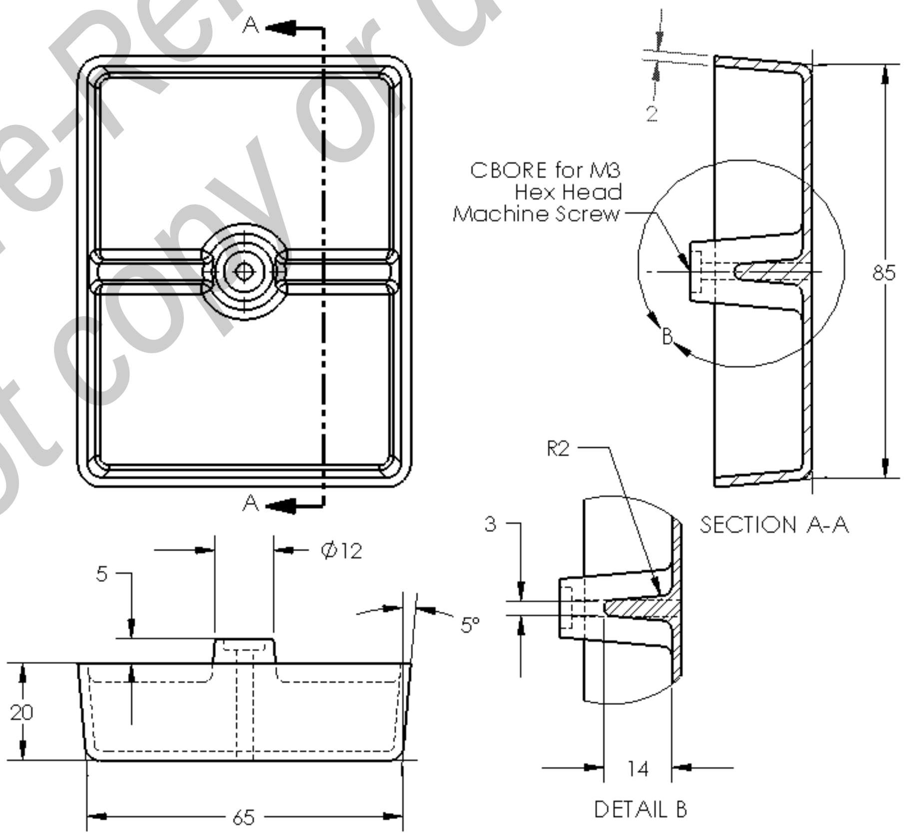
Edit this part using the information and dimensions provided. Use editing techniques to maintain the design intent.

This lab reinforces the following skills:

- **Edit Sketch**.
- **Adding and deleting geometric relations.**
- **Edit Feature**.
- **Edit Sketch Plane**.

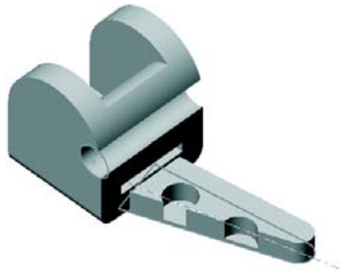
Procedure

Open the existing part `Add Draft`, and make several edits using the final drawing below. Change the model so that **5°** of draft is added.



Exercise 32: Editing

Edit this part using the information and dimensions provided. Use equations, relations or link values to maintain the design intent.



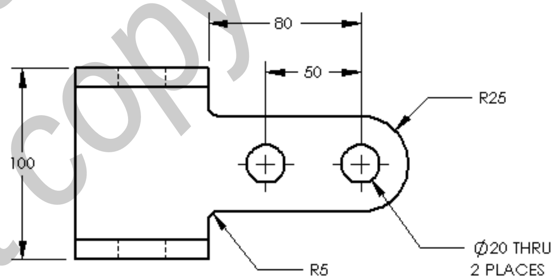

This lab reinforces the following skills:

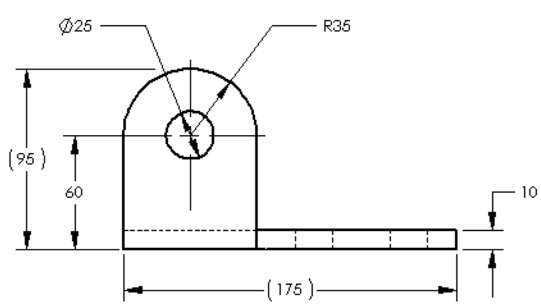
- **Edit Sketch** and **Edit Sketch Plane**.
- Adding and deleting geometric relations.
- **Edit Feature**.
- **Reorder**.
- Inserting Dimensions.

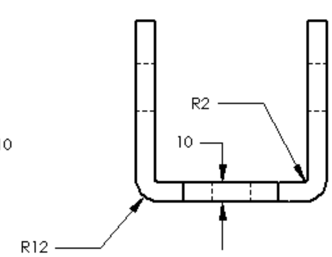
Procedure

Open the existing part `Editing`, and make several edits:

Change the existing part, editing and adding geometry and relations, to match the version shown below.



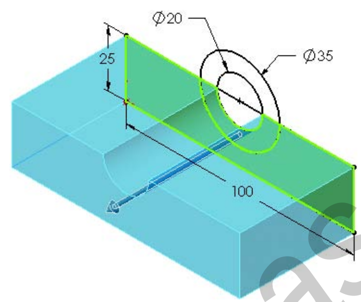
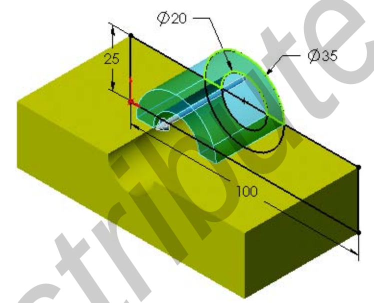
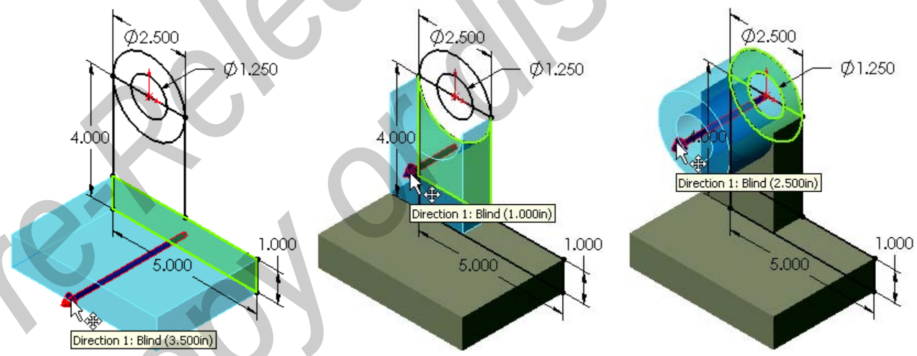
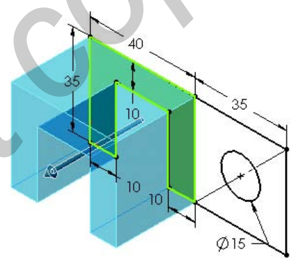
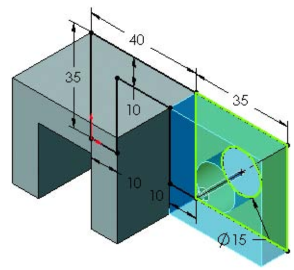
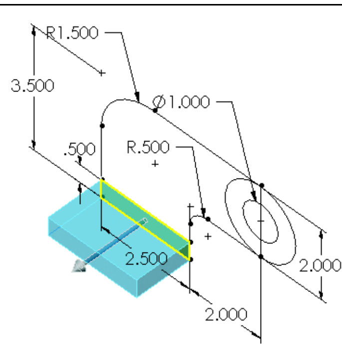
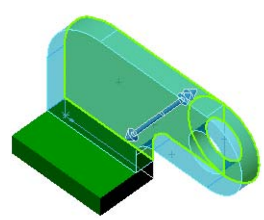




Exercise 33: Contour Sketches #1-#4

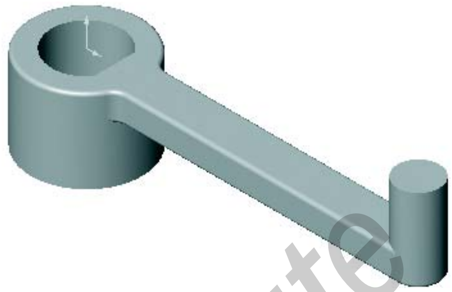
Create this part using the information provided. Extrude profiles to create the parts.

The existing part are `Contour Sketches #1-#4`.

<table>
<tr><td>#1
Depth: 50mm and 30mm</td><td>
</td><td>
</td></tr>
<tr><td>#2
Depth: 3.5”, 1” and 2.5”</td><td colspan="2">
</td></tr>
<tr><td>#3
Depth: 30mm and 10mm</td><td>
</td><td>
</td></tr>
<tr><td>#4
Depth: 1.5” and 0.5”</td><td>
</td><td></td></tr>
</table>

Exercise 34: Handle Arm

Create this part using the information and sketch provided. Extrude profiles to create the part.



This lab reinforces the following skills:

- Contour selection.
- Extrusions.

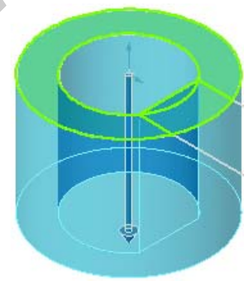
Procedure

Open an existing part.

1 **Open the part named `Handle Arm`.**
It contains a single sketch.

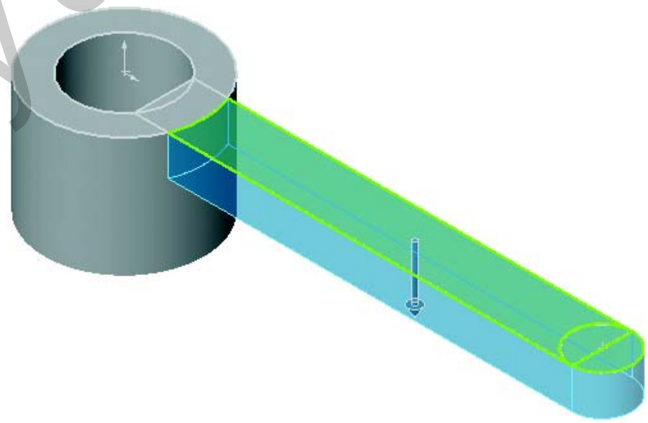
2 **First feature.**
Using the **Contour Select Tool**, select the proper geometry and extrude.

Depth = **0.75”**.



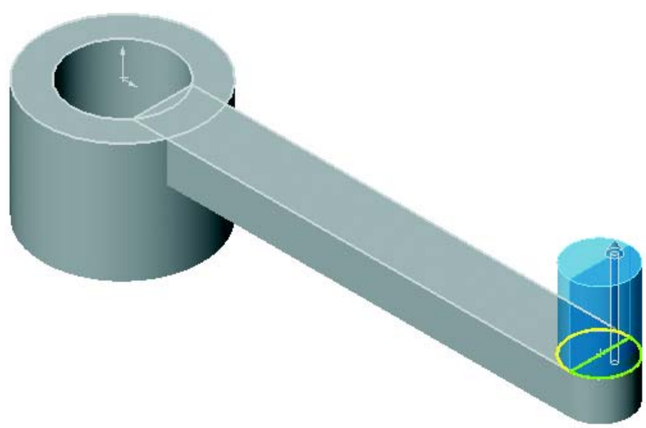
3 **Boss feature.**
Using the *same* sketch, select contours and extrude.

Depth = **0.25”**.

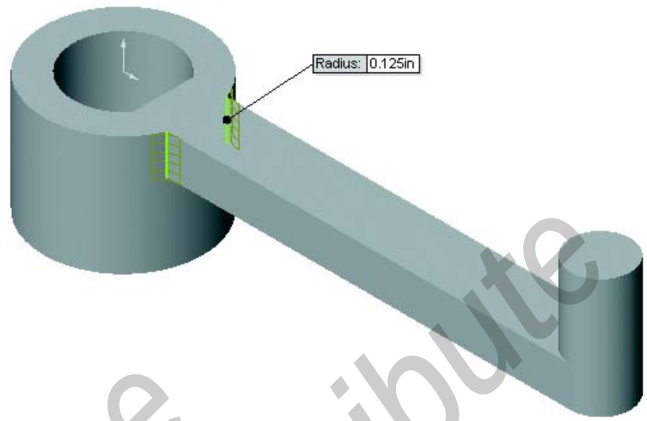


4 **Cylindrical boss.**
Using the *same* sketch, select contours and extrude.

Depth = **0.5”**.

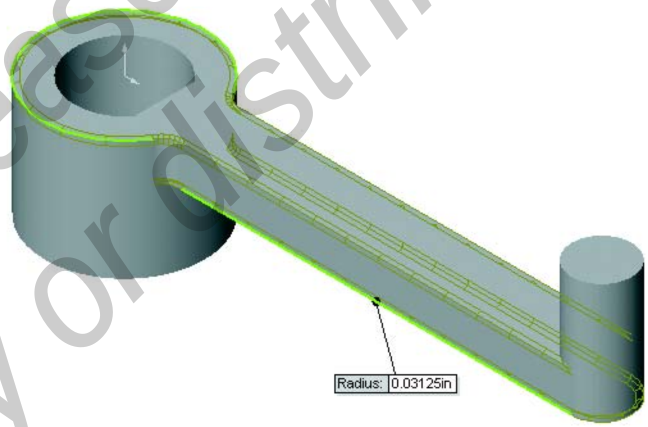


5 Fillets.
Add fillets of radius **1/8”**.



6 Rounds.
Add rounds of radius **1/32”**.

7 Save and close the part.



Exercise 35: Oil Pump

Create this part using the information and sketch provided. Extrude profiles to create the part.

This lab reinforces the following skills:

- Contour selection.
- Extrusions.

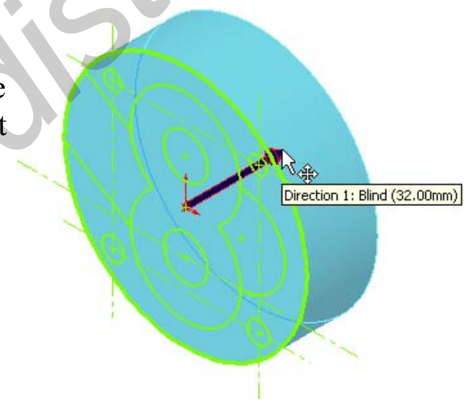
Procedure

Open an existing part.

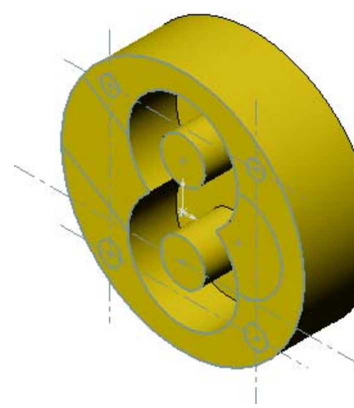
1 **Open the part named `Oil Pump`.**
It contains a single sketch.

2 **Boss feature.**
Use **Contour Select Tool** and select the outermost circle. Extrude it a depth of **32mm** to form the first feature.

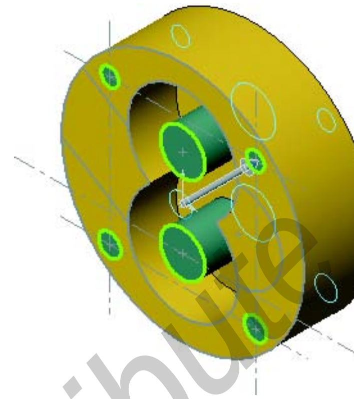
Extrude the boss into the screen so the sketch lies on the front face.



3 **Extrude a cut.**
Select the figure-8 contour and extrude a cut to a depth of **22mm**.



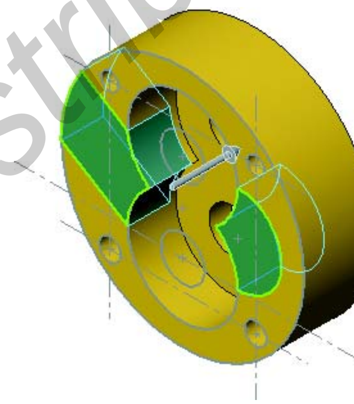
4 Through All cuts.
There are six circles that represent through holes.



Tip To select multiple contours, hold down **Ctrl** and then select each contour.

Question Should these holes be created as a single feature? Or should they be made as three separate features, one for each size hole?

5 Last two cuts.
Extrude the last two contours to a depth of **19mm**.



6 Hide the sketch.

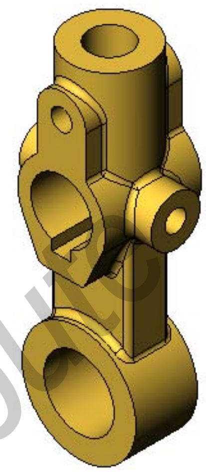
7 Save and close the part.

Exercise 36: Using the Contour Selection Tool

Create this part using the information and sketch provided. Extrude profiles to create the part.

This lab reinforces the following skills:

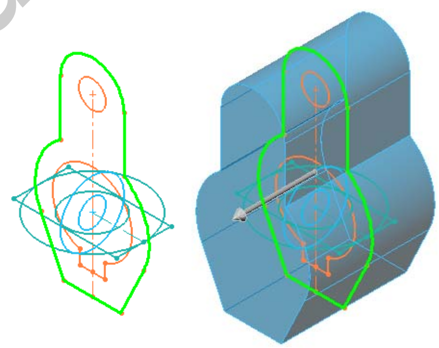
- Contour selection tool.
- Extrusions.
- Fillets.



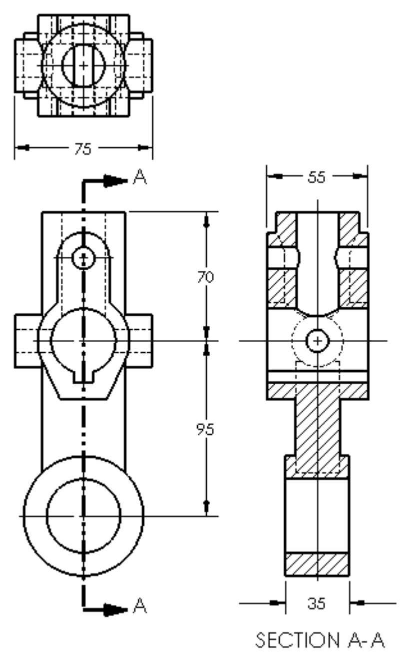
Procedure

Open an existing part.

1. **Open the part named `Idler_Arm_Contour_Selection`.**
   It contains three sketches. Show all sketches.
2. **First feature.**
   Right-click the **Contour Select Tool** and select the indicated geometry.
3. **Extrude.**
   Click the Extrude tool and create the boss using **Mid Plane** and a **Depth 55mm**.



4. **Bosses and cuts.**
   Using the same procedure, create the remaining bosses and cuts as shown.
5. **Fillets and rounds.**
   Use fillets and rounds of **3mm**.
6. **Save and close the part.**



Lesson 9
Configurations of Parts

Upon successful completion of this lesson, you will be able to:

- Use configurations to represent different versions of a part within a single SolidWorks file.
- Suppress and unsuppress features.
- Change dimension values by configuration.
- Suppress features by configuration.
- Understand the ramifications of making changes to parts that have configurations.

Configurations

Configurations allow you to represent more than one version of the part in the same file. For example, by suppressing the machined features (holes, chamfers, pockets, etc.) and changing dimension values in the parts at the top of the illustration, you can represent the rough forgings shown below them.

This lesson addresses the use of configurations in parts. Assembly configurations are covered in another lesson.

Terminology

Some of the terms used when discussing and working with configurations are explained below.

Configuration Name

The **Configuration Name** appears in the ConfigurationManager. It is used to distinguish between configurations within the same part or assembly at the part, assembly or drawing level.

They can be created directly or indirectly through a design table.

Suppress/ Unsuppress Features

Suppress is used to temporarily remove a feature. When a feature is suppressed, the system treats it as if it doesn't exist. That means other features that are dependent on it will be suppressed also. In addition, suppressed features are removed from memory, freeing up system resources. Suppressed features can be unsuppressed at any time.

Other Configurable Items

In addition to features, other items can be suppressed and unsuppressed using configurations:

- **Equations**
- **Sketch Constraints**
- **External Sketch Relations**
- **Sketch Dimensions**
- **Colors**

Sketch Planes and extrude **End Conditions** can be set differently on a configuration by configuration basis.

Using Configurations

Both parts and assemblies can have configurations. Drawings do not have configurations of their own but drawing views can display different configurations of the files they reference.

Design Tables use more automated methods in the creation of configurations. For more information on design tables, see *Design Tables* on page 303.

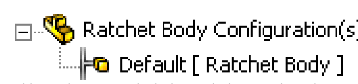
Procedure

In this lesson you will learn about using configurations within a part file. In *Lesson 12: Bottom-Up Assembly Modeling*, you will explore using configurations in conjunction with assemblies.

Begin this example by following this procedure:

1 **Open the `Ratchet Body`.**
This part is a copy of the one created in a previous lesson.

Accessing the Configuration-Manager

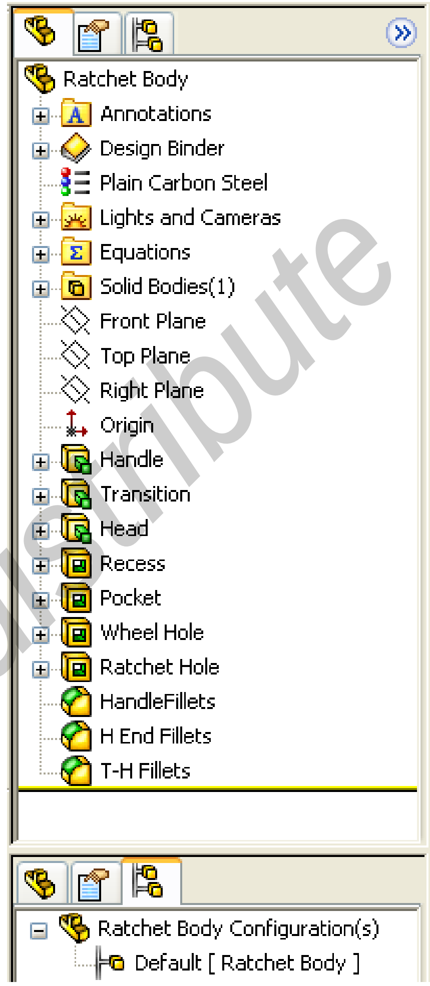


Configurations are managed from within the same window that is occupied by the FeatureManager design tree. To switch the display within this window, use the tabs located at the very top of the window pane. Clicking the tab will display the ConfigurationManager (shown at the upper right) with the default configuration listed. The default configuration is named `Default`. (Who says we don't have a sense of humor?) This configuration represents the part as you modeled it — with nothing suppressed or changed. When you want to switch back to the FeatureManager display, click the tab.

Splitting the FeatureManager Window

Many times it is efficient to be able to access *both* the FeatureManager design tree and the Configuration-Manager at the same time. This is particularly true when working with configurations. Rather than switch back and forth using the tabs, you can split the FeatureManager window top to bottom, creating two panes. One pane can show the FeatureManager design tree and the other can show the Configuration-Manager.

To subdivide the FeatureManager window into two panes, drag the splitter bar downwards from the top of the window. Use the tabs to control what is displayed in each pane.



Adding New Configurations

Every part (and assembly) must have at least one configuration, and multiple configurations are common. There are several options beyond the **Configuration name** that you can set.

Bill of Materials Options

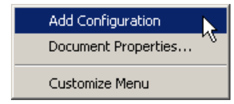
When the part is used in an assembly and further, a bill of materials, set the name that should appear under Part Number.

Advanced Options

The advanced options include rules for creation of new features and color settings. Parent/Child options are for assemblies only.

- **Suppress Features**
  This option controls what happens to newly created features when other configurations are *active* and this configuration is *inactive*. If checked, new features added with other configurations active are suppressed in this one.
- **Use configuration specific color**
  Allows for different colors for each configuration using the color palette. Different materials may introduce different colors.

2 Adding a new configuration.
Position the cursor within the ConfigurationManager and from the right-mouse menu, choose **Add Configuration...**

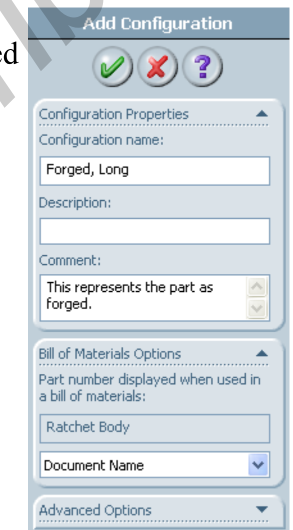


When you add a configuration that configuration becomes active. Any subsequent changes to the part (such as suppressing features) are stored as part of the configuration.

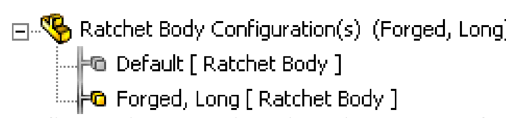
Tip Special characters such as the slash (/) are not allowed in the configuration name.

3 Add configuration.
The **Add Configuration** property manager is used to add configurations to the part. Give the configuration the name `Forged, Long` and optionally, add a comment.

Click **OK**.



4 Added to list.
The new configuration is added to the list and automatically made the active configuration. Notice that the name of the active configuration is shown in parentheses, appended to the part name icon.



Defining the Configuration

You define the configuration by turning off or suppressing selected features in the part. When a feature is suppressed, it still appears in the FeatureManager design tree but it is grayed out. This version of the part is saved or stored in the active configuration. You can create many different configurations within a part.You can then easily switch between different configurations using the **ConfigurationManager**.

Introducing: Parent/Child Relationships

Parent/Child is used to display the dependencies between features. Both the features it is dependent upon (Parents) and the features which depend upon it (Children) are displayed.

Where to Find It

- From the right-mouse button over a feature click **Parent/Child**.

Introducing: Suppress

Suppress is used to remove a feature from memory, essentially deleting it from the model. It is used to remove selected features from the model to create different “versions” of that model. All the children of a feature that is suppressed are suppressed with it.

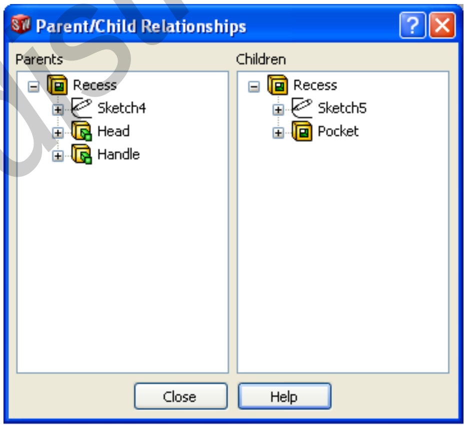
Unsuppress and **Unsuppress with Dependents** are used to reverse the effect of suppression on one (unsuppress) or more (unsuppress with dependents) features.

Where to Find It

- From the right-mouse menu click **Suppress**.
- Or click the **Suppress** tool on the Features toolbar.
- Or choose **Edit, Suppress** from the pulldown menu.
- Or click **Suppressed** in the **Feature Properties** dialog box.

---

5 Check Parent/Child.
Right-click the `Recess` feature and select **Parent/Child**. Expand the `Pocket` feature on the **Children** side to see other child features.



6 Suppress the `Recess` feature.
In the FeatureManager design tree, select the `Recess` feature. Right-click **Suppress**.

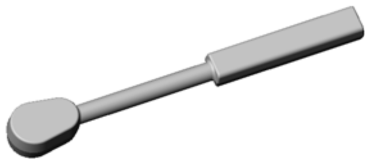
The system suppresses not only the `Recess` but also the `Pocket`, the `Wheel Hole`, and the `Ratchet Hole`. Why?

Because the `Pocket`, `Wheel Hole`, and `Ratchet Hole` are all children of the `Recess`. If you recall, the `Pocket` was sketched on the bottom face of the `Recess`. The two holes were then sketched on the bottom face of the `Pocket`. This is what established the parent-child relationships among them.

Rule

Suppressing a feature automatically suppresses its children.

When the features are suppressed in the FeatureManager design tree, their corresponding geometry is suppressed in the model, too.

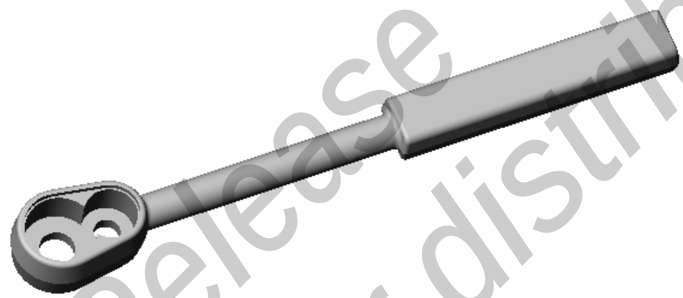


Changing Configurations

To switch to a different configuration, simply double-click on the one that you want.

7 Switch back to the `Default` configuration.
Position the cursor over the `Default` configuration icon and double-click it.

The system keeps the `Recess`, `Pocket`, `Wheel Hole`, and `Ratchet Hole` features unsuppressed making them visible in both the FeatureManager design tree and the graphics windows.



Renaming and Copying Configurations

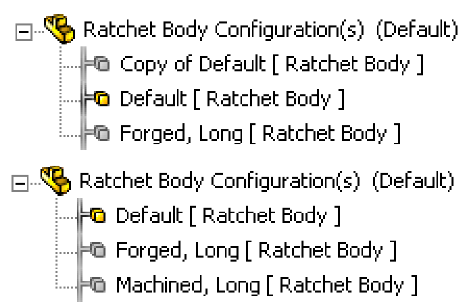
We now have two configurations: `Default` and `Forged, Long`. The `Default` configuration represents the part in its machined state. However, the name `Default` is not too meaningful.

Configurations can be renamed in the same way as features. However, if a configuration is being referenced by another SolidWorks document, renaming that configuration can cause some difficulties.

A Better Approach

Instead of renaming the default configuration we will make a copy of it and then rename the copy.

8 Copy the `Default` configuration.
Select the `Default` configuration and copy it using any of the standard techniques for copying a feature: **Ctrl+C**, **Edit, Copy**, or the tool.



Paste the configuration using **Ctrl+V**, **Edit, Paste**, or the tool.

Rename the copy to `Machined, Long`.

You now have configurations that represent the `Ratchet Body` in its forged and machined states.

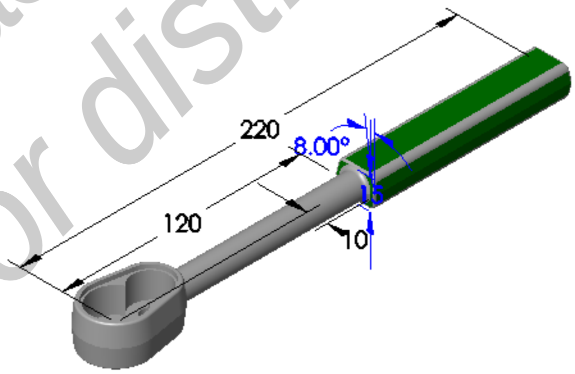
9 Create more configurations.
Using the same procedure, copy and paste the `Forged, Long` configuration. Rename it `Forged, Short`. Copy and paste the `Machined, Long` configuration, renaming it `Machined, Short`.

Changing Dimension Values

Configurations can also be used to control the value of a dimension. Each configuration can be used to change the dimension to a different value. The change can be configured for the active, specified, or all configurations.

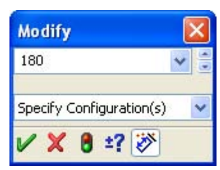
In this example, the “`Short`” configurations will have a slightly shorter handle length.

10 Key dimension.
Double-click `Machined, Short` to make it the active configuration. Double-click the `Handle` feature to expose the sketch dimensions.

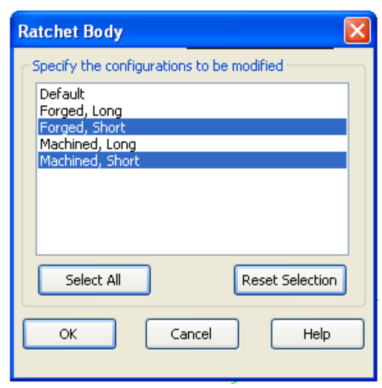


Configure the dimension.

Double-click the 220mm dimension and change it to **180mm**. In the dropdown, choose **Specify Configuration(s)**. Click **OK**.



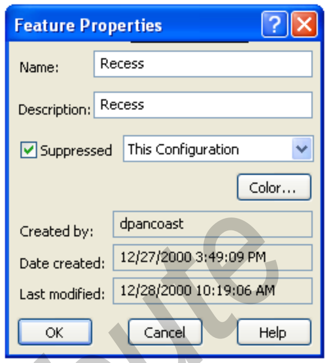
Select *only* the `Forged, Short` and `Machined, Short` configurations from the list and click **OK**.



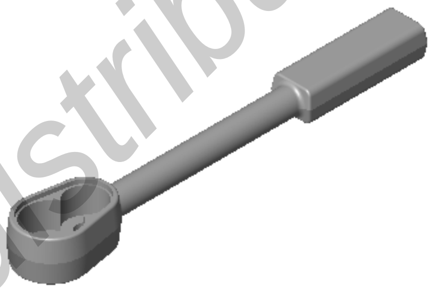
Note

Features can be suppressed or unsuppressed in the active, specified, or all configurations using **Properties**.

Right-click the feature and select **Feature, Properties**. Check or clear **Suppressed** and select the configurations using the dropdown list.



11 Changes.
Rebuild the model to see the changes in the current configuration.



Editing Parts that Have Configurations

When configurations are added to a part, features may be automatically suppressed, dialogs list many additional options, and other strange things can happen. This section shows what happens when there are multiple configurations in the part being edited.

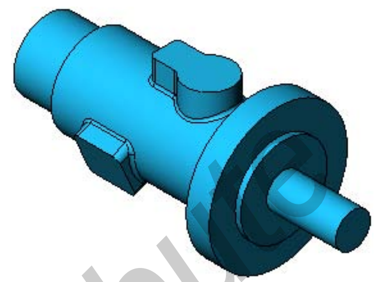
Every configuration in a part contains the same features. However, in different configurations, those features may have different suppression states, different dimension values, different end conditions, or even different sketch planes.

The table below list options that can be controlled using configurations.

Type	**Possibilities**
Feature	Suppress/Unsuppress by configuration
Equations	Suppress/Unsuppress by configuration
Sketch Relations	Suppress/Unsuppress by configuration
External Sketch Relations	Suppress/Unsuppress by configuration
Sketch Dimensions	Suppress/Unsuppress by configuration
End Conditions	Different end condition for each configuration
Sketch Planes	Different sketch plane by configuration

Note Colors, textures and materials can also be configured.

1. **Open the part.**
Open the part `WorkingConfigs`. This part has one configuration: `Default`. Configurations and new features will be added to the part.

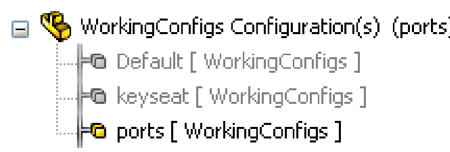


2. **Creating new configurations.**
Switch to the Configuration Manager and right-click **Add Configuration**. Create a new configuration named `keyseat`.

3. **Copy and paste.**
Copy and paste `keyseat` to create another new configuration. Name it `ports` and make it the active configuration.

   By default the option **Suppress features** is selected. This means that as new features are added, they are suppressed in all configurations except the active one.

4. **Active configuration.**
Make sure the configuration named `ports` is active. At this time, all three configurations are the same.



Tip Copy and paste makes a duplicate of the copied configuration. Note that the name in square brackets is the name that will appear in a BOM. This can be changed by changing the setting for the **Part number displayed when used in a bill of materials** in the **Configuration Properties** dialog.

Design Library

The **Design Library** is a collection of features, parts and assembly files within the **Task Pane**. The files can be inserted into parts and assemblies to reuse existing data. The `features` folder will be used in this example.

The Features Folder

The `features` folder contains **Library Feature** (`*.sldlfp`) files. They can be added to a part by simply dragging and dropping them onto a planar face of the model. References needed by the feature are selected, attaching dimensions and relations. The references are followed by position and size change options to set the values of the dimensions in the feature.

Default Settings

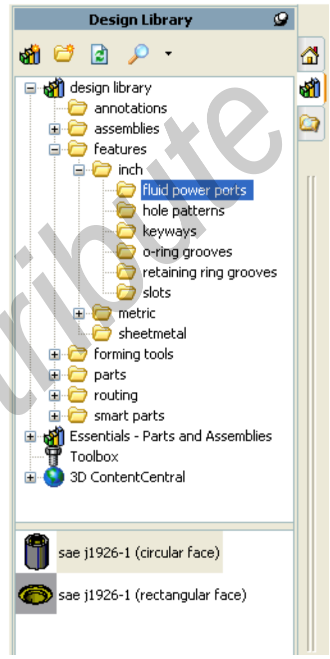
The first of three library features will be inserted using the default settings for location and size.

5 Folders.
Click the **Design Library** and the pushpin.

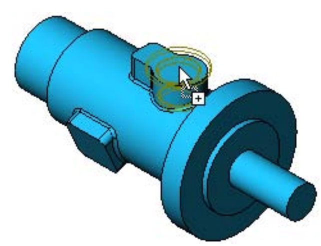
Expand the `features` folder.

Expand the `inch` folder.

Click the `fluid power ports` folder.



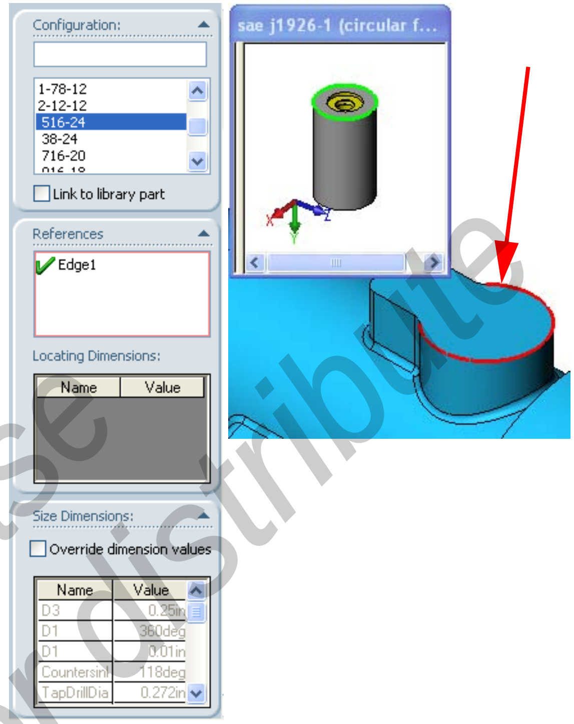
6 Drag and drop.
Drag and drop the `sae j1926-1 (circular face)` feature onto the planar model face as shown. The drop face is the **Placement Plane** for the feature.



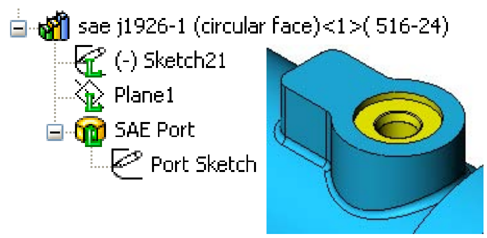
7 Settings and selections.
Select the configuration `516-24` from the list. Select the `Edge1` (circular edge) reference as indicated in the preview window.

The **Link to library part** option will create a link to update this part from changes in the library feature.

Checking **Override dimension values** allows the internal dimension values of the feature to be changed.



8 Feature.
The library feature is added to the FeatureManager as a library feature consisting of sketches, a plane and a cut.



Note The “**L**” labels superimposed over the feature icons indicate a library feature.

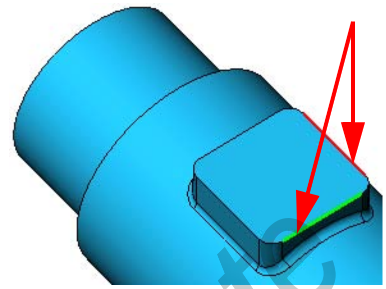
Multiple References

Many features contain multiple references to faces, edges or planes. These are used to attach dimensions and set relations on geometry.

If the references are not properly attached to model geometry, they will become dangling. For more information, see *Reattach Dimensions* on page 230.

9 References.

Drag and drop the `sae j1926-1 (rectangular face)` feature onto the planar face. This feature requires the selection of two references, each being a linear model edge.

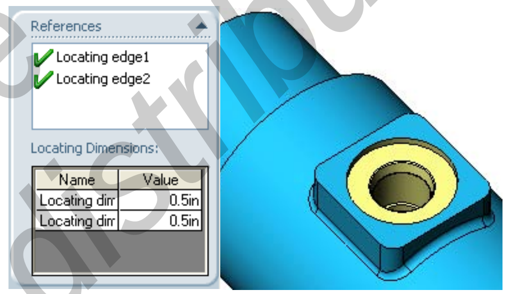


Select the `716-20` configuration. For the two **References**, select the shown in green edge followed by the red.

10 Dimension values.

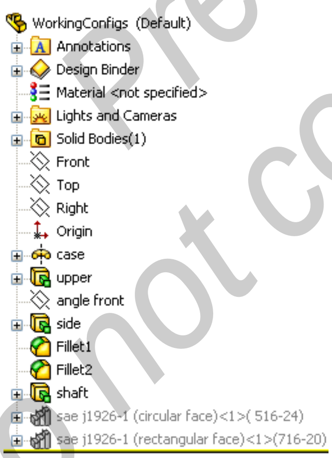
Set each **Locating Dimension** to **0.5"** by clicking the cell and typing.

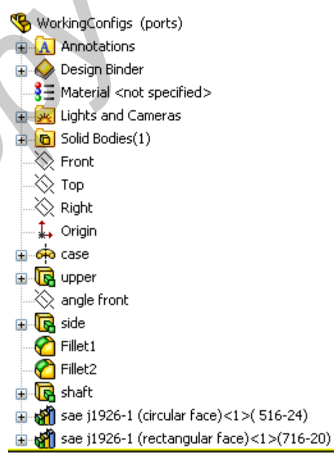
Click **OK**.

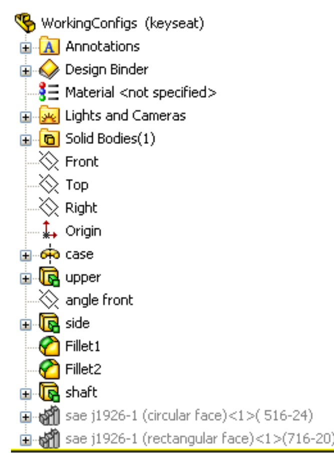


11 Check configurations.

The new features are *unsuppressed* in the active configuration (`ports`) but *suppressed* in all the others.







12 Active configuration.

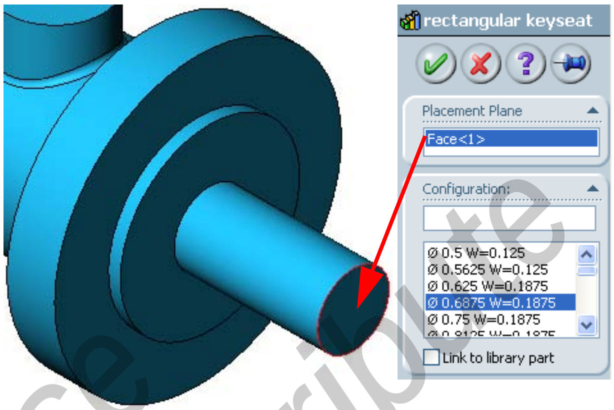
Make the configuration `keyseat` the active one.

Dropping on Circular Faces

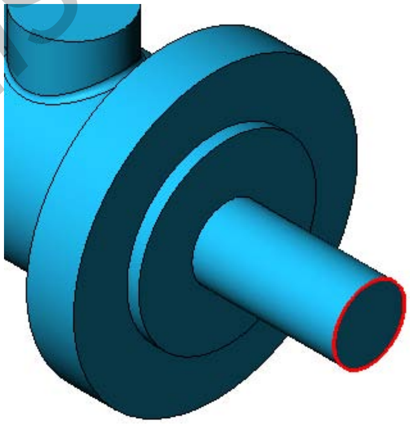
Some features attach to circular faces of the target model and require the first "drop" face to be that face. In these cases, the Placement Plane is selected after the drop.

13 Feature.
Open the `keyways` folder in the design library. Drag and drop the `rectangular keyseat` feature onto the *circular* face of the shaft.

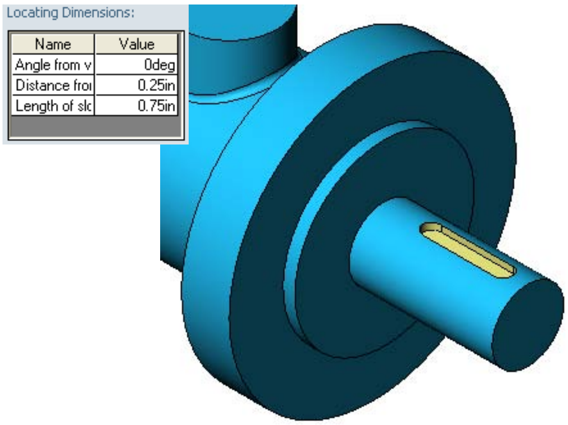
Select ∅ `0.6875 W=0.1875` configuration and the planar end face as the **Placement Plane**.



14 Reference.
Select the circular *edge* of the end face as the **Reference**.

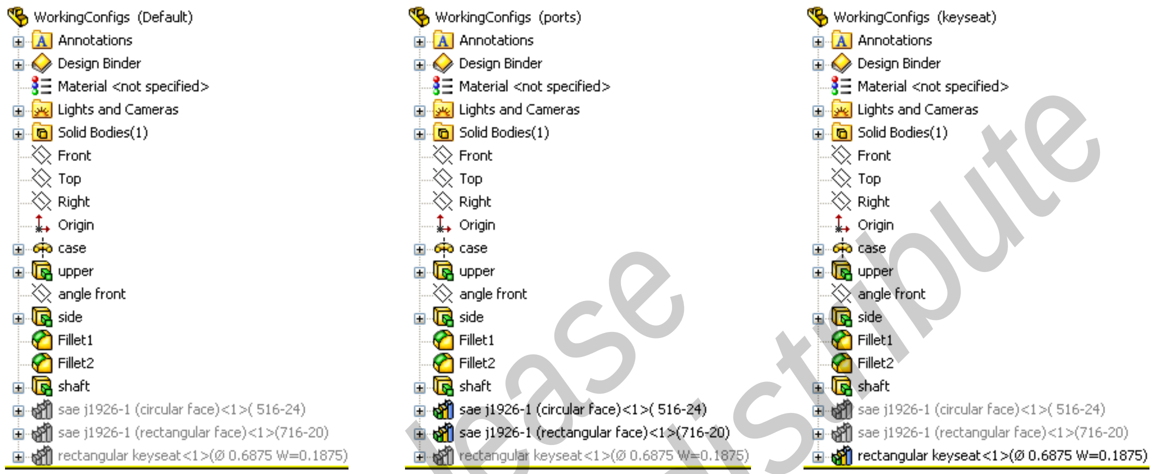


15 Locating dimensions.
Set the **Locating Dimensions** to the values shown.



16 Check configurations.

The new feature is *unsuppressed* in the active configuration (`keyseat`) but *suppressed* in all the others.



17 Save and close the file.

Exercise 37: Configurations

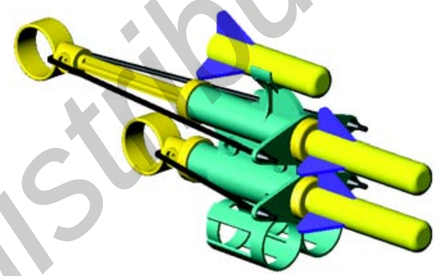
Use an existing part as the basis for a series of configurations. Create different versions by suppressing various features in each configuration.

This lab reinforces the following skills:

- Creating configurations.
- Suppressing features.

What is this Thing?

The part used in this example is the main twin barrel component from a toy that shoots soft, foam rockets.

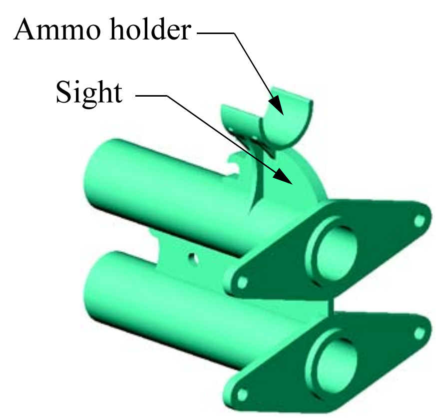


Procedure

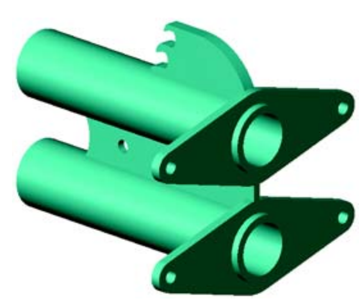
Open the existing part `config part`.

1 Create new configurations.
Manually create new configurations to match the conditions and names below. Add features to the model where required.

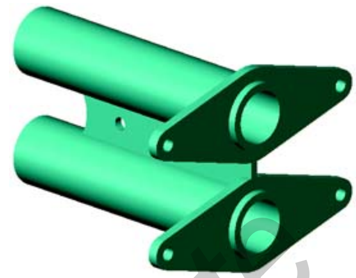
- `Best` model – Includes ammo holder and sight.



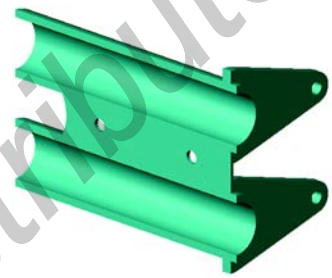
- `Better` model – Includes sight only.



- `Standard` model – Includes neither ammo holder nor sight.



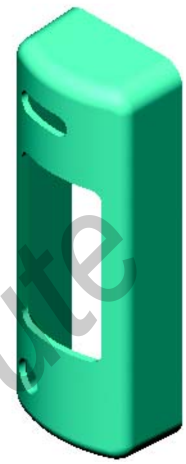
- `Section` model – Shows a section cut through the `Standard` model.



Note The `Section` configuration is created using a cut feature. To create the cut feature activate the `Standard` configuration. Then use the `Front` reference plane and the command **Insert, Cut, With Surface...**, cut the model.

2 Save and close the part.

Exercise 38: More Configurations



Use an existing part and create configurations. Create new features which are controlled by the configurations.

This lab reinforces the following skills:

- Adding configurations.
- Suppressing features.

Configurations allow the shape of the part to differ based on which features are displayed.

100 Series		**200 Series**	
100CF	100SF	200CF	200SF
			

Procedure

Open the existing part `Speaker`.

1 **Create new configurations.**
Using the names below, create four new configurations in the part. They represent two variations of the speaker and its function. (`C` = Control, `S` = Slave and `F` = Front).

- `100CF`
- `100SF`
- `200CF`
- `200SF`

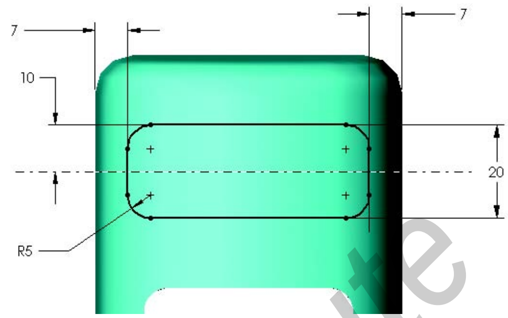
2 **Configuration settings for `volume control`.**
Add the feature `volume control`. Suppress and/or unsuppress the feature to get the shape shown here for the configurations listed above.

3 **Configuration settings for `rounded tweeter`.**
Add the feature `rounded tweeter`. Suppress the feature in both `200` series configurations.

4 **Configuration `200CF`.**
Switch to the configuration `200CF` in the ConfigurationManager.

5 New cut.

Right-click the sketch `opening locations`, and select **Show**.



Add a new cut using the geometry shown. Set the dimension values for **All Configurations**.

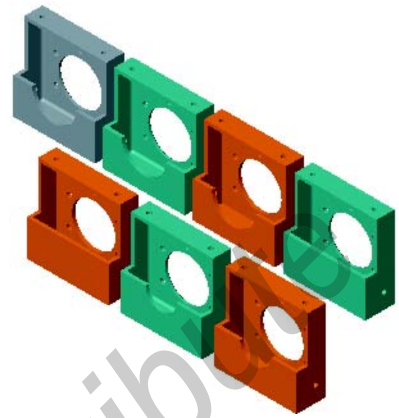
Rename the feature `tweeter`.

Set the state to suppressed for the `100` series and unsuppressed for the `200` series configurations.

6 Save and close the part.

Exercise 39: Working with Configurations

Using an existing part with configurations, add new features and modify others.



This lab reinforces the following skills:

- Suppressing a feature using feature's properties.
- Editing extrusion end conditions by configuration.
- Adding new features in a multi-configuration environment.

Procedure

Open the existing part `Working with Configurations`.

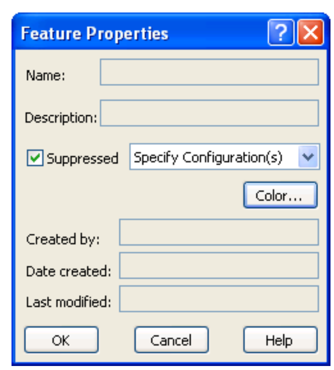
1 **Configurations.**
The part contains seven (7) configurations. They are:

- `Default`
- `doublewall`
- `doublewall.simple`
- `singlewall`
- `singlewall.simple`
- `stepped`
- `stepped.simple`

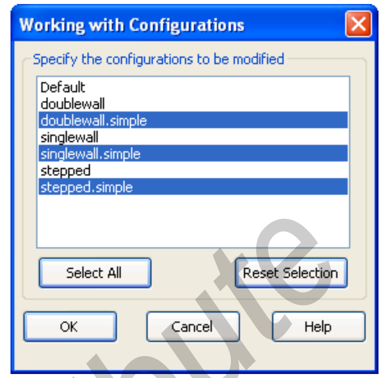
To begin with, they are all the same – copies of the `Default` configuration.

2 **Select fillet and chamfer features.**
Modify the three simplified configurations to suppress all fillets and chamfers within them. Press **Shift** and select the features `fill060.vert`, `fill060.tb`, and `cham025`.

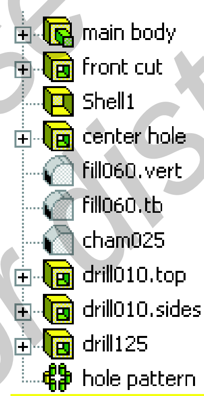
3 **Feature Properties.**
Click **Edit, Properties**. Click **Suppressed**, **Specify Configuration(s)** and click **OK**.

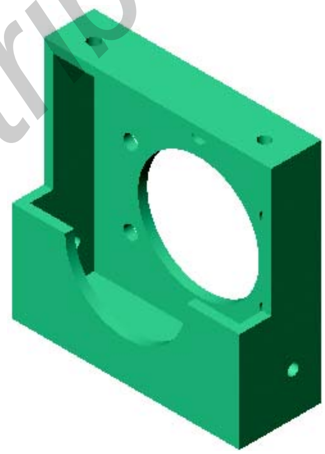


4 Configuration selection.
Select `doublewall.simple`, `singlewall.simple` and `stepped.simple` from the list. Deselect all others and click **OK**.



5 Changes.
The three features are suppressed in only the selected (simplified) configurations.





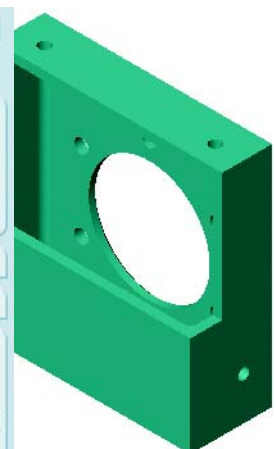
6 Active configuration.
Make `singlewall` the active configuration.

7 Modify end conditions.
Edit the definition of the `center hole` feature.

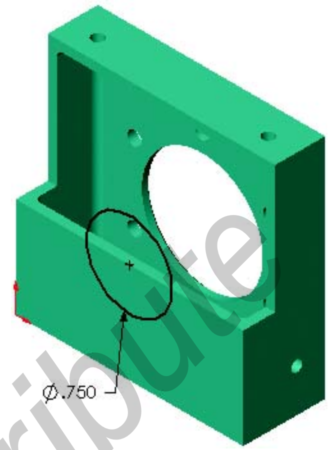
Set the **Configurations** to **Specify Configurations** and choose:

- `singlewall`
- `singlewall.simple`
- `stepped`
- `stepped.simple`

Choose **Up To Next** as the end condition and click **OK**.

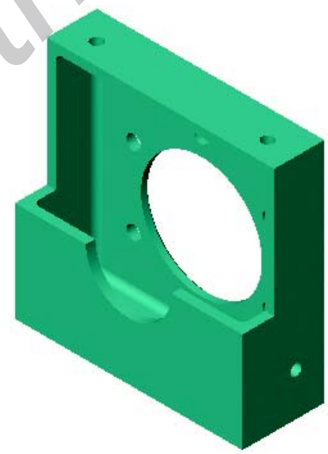


8 Add a new feature.
Set the active configuration to `stepped`.



Create a new sketch on the front face of the model and create a circle concentric to `center hole`. Add a dimension of **0.75”** for **All Configurations**.

9 Cut.
Create a cut feature using the **Up To Next** end condition. The feature is added to all configurations, suppressed in all but the active one.



Name the feature `step cut`.

10 Feature Properties.
Right-click the new feature and select **Feature Properties...**. Make sure the **Suppressed** check box is *cleared*.

Select **Specify Configuration(s)** and click **OK**. Select the configurations `stepped` and `stepped.simple` and click **OK**.

Tip The configurations can also be differentiated by color. Use the **Configurations** group box on the **Edit Color** dialog.

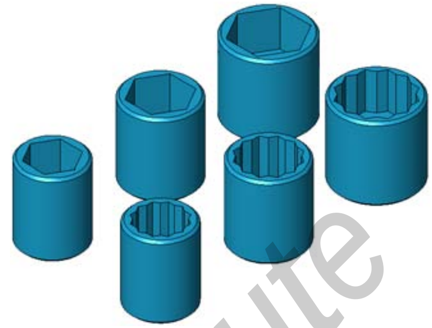
11 Save and close the part.

Lesson 10
Design Tables and Equations

Upon successful completion of this lesson, you will be able to:

- Link dimension values together to capture design intent.
- Create Equations.
- Automatically create design tables.
- Use existing design tables to create families of parts.
- Make detail drawings using more advanced types of drawing views.

Design Tables



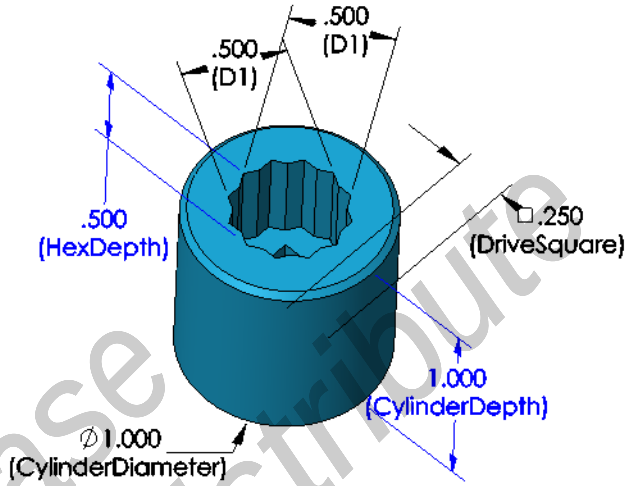
Design Tables are the best way to create configurations of parts. They are used to control dimension values for families of parts, and suppression states of features. Design tables can be used to create a family of parts from a single part design. Since the SolidWorks software is an OLE/2 application, an Excel spreadsheet is used to lay out the design table so it can be imported into the SolidWorks document.

Key Topics

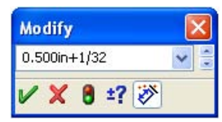
The key topics covered in this lesson are shown in the following list.

- **Link Values**
  Link values are used to set two or more dimensions equal.
- **Equations**
  Equations can be used to create algebraic relationships between dimensions using mathematical operators and functions.
- **Auto-create a Design Table**
  Design tables can be automatically created and edited after they are inserted. Design tables can be set so that you cannot change the model if those changes would update the design table.
- **Making changes**
  Changes can be made to the existing design table by editing the table to add configurations, dimensions or features.
- **Bi-directional Changes**
  Dimensions that appear in the design table are driven by it. Changes made to the dimensions in the model force the corresponding change in the design table.
- **Inserting Blank Design Tables**
  Blank design tables are useful for many purposes including exploded views and multiple positions of a component in an assembly.
- **Drawings with Configurations**
  Drawings with parts that have configurations provide many options for display. Any available configuration can be displayed in a model view.

1 Open `Socket`.
The `Socket` part (shown here with **Show Feature Dimensions** and **Show Dimensions Names** on) contains two cut features that represent the overlapping hexagon cuts. They retain their original names.



2 Change value.
Double-click the sketch of the `6 Point` feature and double-click the **0.5”** dimension. Add **1/32”** to the nominal value as shown. Rebuild the model.



Link Values

Link Values can be used to set a series of dimensions equal by assigning them the same name. Changing the value of any of the linked dimensions changes all of them. The linking can be removed using **Unlink Value**. This option is superior to equations for setting several values equal to each other.

In this example there are two linear dimensions: one in each of the hexagon shaped cuts. Link values will be used to tie them together.

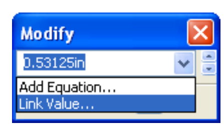
Where to Find It

- From the Dimension Modify dialog, choose **Link Value**.
- Right-click one or more dimensions and select **Link Values**.

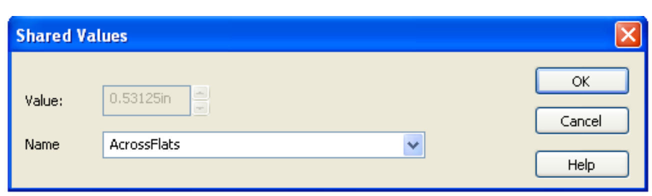
Note

The dimensions being linked together must be of the same type. Link angular dimensions to other angular dimensions and so forth.

3 Access link values.
Double-click the same dimension as if to change the value. Using the dropdown menu, select **Link Value**.

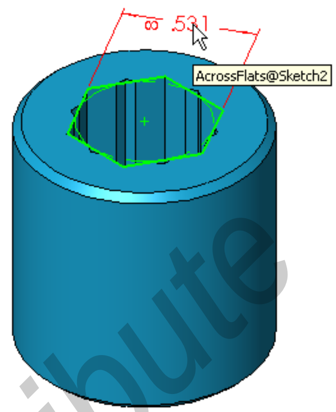


4 Name the link value.
In the **Shared Values** dialog, type the name `AcrossFlats` and click **OK**.

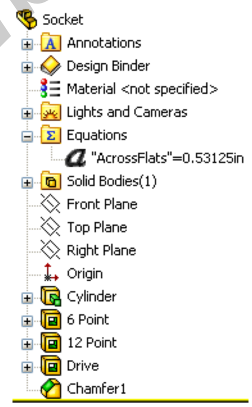


5 Link Value added.
The link value is added and is used as the dimension name. A prefix symbol is also added to identify this dimension as being linked.

Rebuild the model.



6 Equation folder.
The link value is listed under the `Equations` folder in the FeatureManager.



7 Add link value.
Double-click the sketch of the `12 Point` feature and double-click the **0.5”** dimension. Select the link value `AcrossFlats` from the dropdown. The value of the existing link value is assigned to this dimension. Rebuild the model.

Note A link value will remain attached to a dimension unless it is removed using right-click **Unlink Value**. Changing the value of *any* linked dimension changes *all* linked dimensions.

Equations

Many times you will need to establish a relationship between parameters that cannot be achieved using geometric relations or modeling techniques.

For example, you can use equations to establish mathematical relations between dimensions in the model. This is what we will do next.

This equation will establish a relationship between the diameter of the cylinder and the distance across the flats of the hex. As the distance across the flats increases, so will the diameter.

Note Simple equality statements *within a part* can be created more easily with **Link Values** than equations.

Preparation for Equations

Although you can begin writing equations and applying them to the model with little or no preparation, it is a much better practice to make a small investment in time up front to achieve added benefit later on. You should consider the following:

- **Renaming the dimensions**
  Dimensions are created by the system with somewhat cryptic default names. To make it easier for others to interpret the equations and understand what exactly is being controlled by them, you should rename the dimensions giving them more logical and easily understood names. Right-click a dimension and choose **Properties** to rename it.

Note

When equations are used in an assembly, the full name uses the form: `Name@FeatureName@PartName`.

- **Dependent versus independent**
  The SolidWorks software uses equations of the form *Dependent = Independent*. This means that in the equation $A = B$, the system solves for A when given B. You can edit B directly and change it. Once the equation is written and applied, you cannot directly change A. Before you start writing equations, you need to decide which parameter will *drive* the equation (the independent one) and which will be *driven* by the equation (the dependent one).
- **Which dimension drives the design?**
  In this example, we will control the diameter of the cylinder based on the distance across the flats of the hex. This means the flat distance is the *driving* or *independent* parameter and the diameter is the *driven* or *dependent* one. The size of the hex drives the design.

Functions

The functions displayed as buttons on the **Add Equation** dialog box include basic operators, trig functions and many more.

Equation form

The equation required in this example uses the distance across the flats of the hex as the driving dimension. This forces changes in the cylinder diameter, a feature that *precedes* it. The form is:

Driven Dimension = Driving Dimension + Constant

where:

Driven Dimension = `CylinderDiameter@Sketch1`

Driving Dimension = `AcrossFlats@Sketch2`

Constant = `0.25`

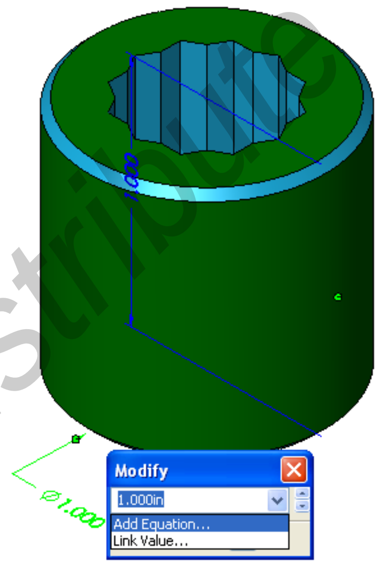
Introducing: Equations

The **Equations** dialog can be used to add, edit, delete and configure equations.

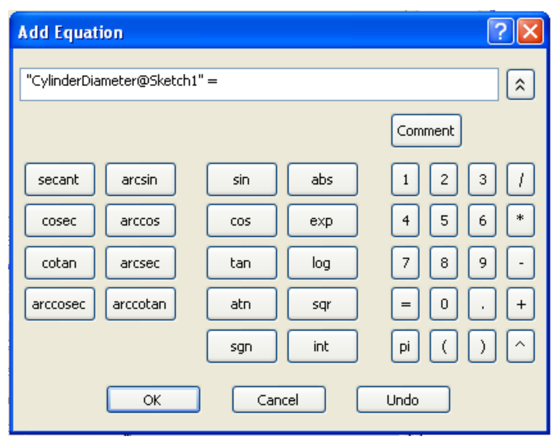
Where to Find It

- Click **Equations** on the Tools toolbar.
- Or from the **Tools** menu, click **Equations**.
- Or right-click the `Equations` folder and choose an option.
- Or from the Dimension Modify dialog, choose **Add Equation**.

8 Add Equation.
Double-click the `Cylinder` feature and the diameter dimension (**1”**). In the dialog, choose **Add Equation** from the dropdown list.

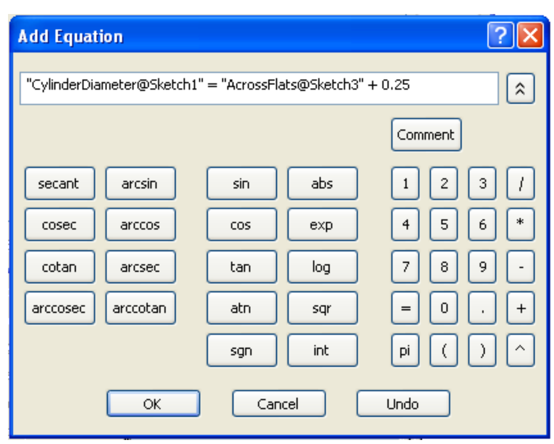


9 Dimension added.
The dimension is added to the new equation on the *left* side of the equals sign.



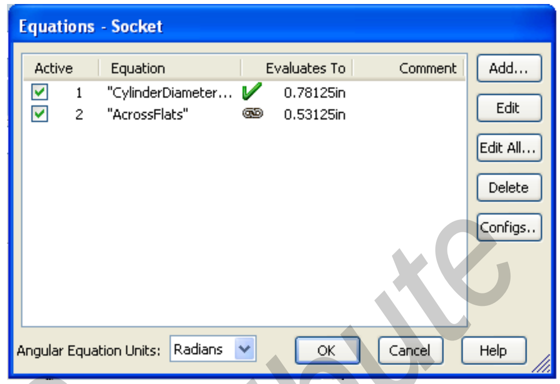
10 Complete equation.
Click either of the link value dimensions and add “**+ 0.25**” to complete the equation.

Click **OK** to add the equation.

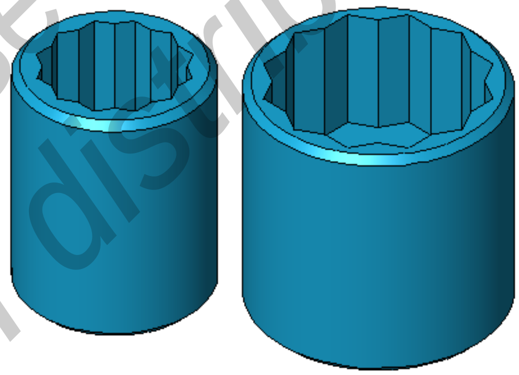


11 List.

The current list of equations, including link values , is listed in the **Equations** dialog. Click **OK** and rebuild the model.

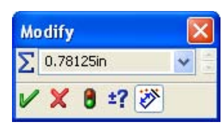


The **Evaluates To** column refers to the value of the `CylinderDiameter@Sketch1` dimension. Changes to the `AcrossFlats@Sketch2` dimension force the evaluation to change.

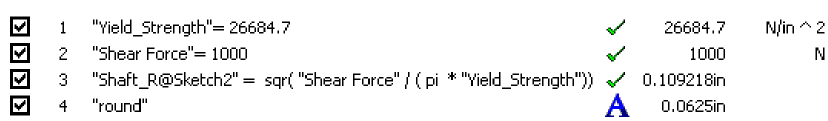


Tip If your equations make use of angular dimensions, select **Radians** or **Degrees** as the **Angular Equation Units**.

Note The driven dimension, `CylinderDiameter@Sketch1` in this case, cannot be changed directly. Double-clicking it leads to a grayed out Modify dialog.

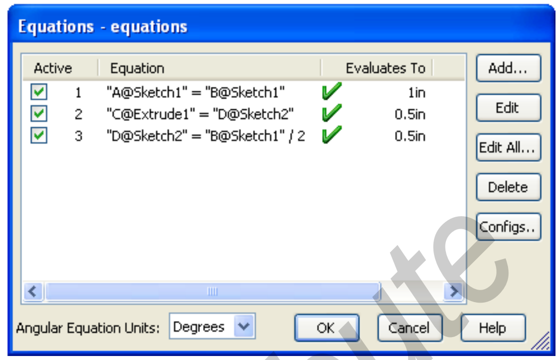


Global Variables **Global Variables** (or independent variables) can be added and used within equations to represent yield strength, poissons ratio or other constants. They can be used within equations.



A Few Final Words About Equations

Equations are solved in the order in which they are listed. If you change a dimension and discover that it takes *two* rebuilds to update all of the part's geometry, this may indicate that your equations are in the wrong order. Edit the equations and use the list to reorder them. Consider this example:



Given three equations: *A*=*B*, *C*=*D*, and *D*=*B*/2, consider what happens if you change the value of *B*. First, the system will compute a new value for *A*. When it evaluates the second equation, nothing is changed. When the third equation is evaluated, the changed value of *B* yields a new value for *D*. However, it isn't until the second rebuild that this new value for *D* gets used to compute a new value for *C*. Reordering the equations thus: *A*=*B*, *D*=*B*/2, and *C*=*D* solves the problem.

Design Tables

Design Tables are the best way to create configurations of parts. They are used to control dimension values for families of parts and suppression states of features.

Auto-create a Design Table

The easiest way to create a **Design Table** in a part is to **Auto-create** it using existing configurations, dimensions and features. The existing information is automatically formatted into an Excel spreadsheet and updates automatically, by default bi-directionally.

Introducing: Insert Design Table

When you insert a design table, the SolidWorks window switches to Excel. That is, the toolbars become Excel toolbars instead of SolidWorks toolbars while the table is active.

There can only be *one* design table in each part. It is stored within the part document unless a linked design table is used.

Where to Find It

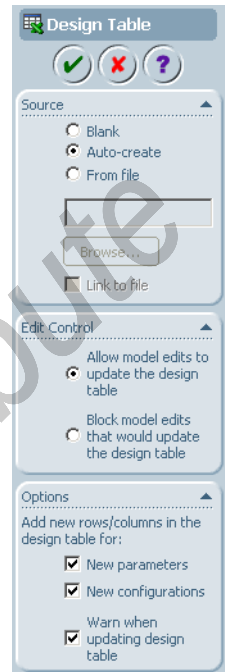
- Click **Insert, Design Table...**.
- Or, click **Design Table** on the Tools toolbar.

FeatureManager Design Tree

When a design table is added to a part or assembly, this symbol Design Table appears in the FeatureManager design tree.

12 Insert a new design table.

Click **Insert, Design Table....** For the **Source** select **Auto-create** to automate the creation of the design table.



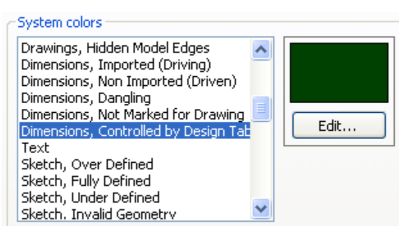
- **Source** determines where information comes from. **Blank** creates a new, blank, design table. Dimensions and features can be added to the design table by double-clicking them. **Auto-create** generates a design table from existing configurations, dimensions and features. **From file** imports an existing Microsoft Excel spreadsheet to use as the design table. Click **Browse** to locate the table. You can also select the **Link to file** check box, which links the spreadsheet to the model.
- **Edit Control** controls the ability to make bi-directional changes. Selecting **Allow model edits to update the design table** means that changes can be made outside the table and it will still be updated.
- **Options** determines how new information in treated by the design table. If it is set to bi-directional, new features, configurations and dimensions changes force updates in the design table.

Tip

Dimensions that are driven by the design table appear in a different color scheme. You can change the color of dimensions that are controlled by the design table to make them easier to identify.

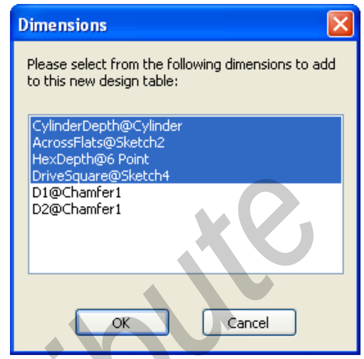
Click **Tools, Options, System Options, Colors**.

Select the option **Dimensions, Controlled by Design Table** and edit the color.



13 Dimensions to add.

The **Auto-create** option generates a list of the dimensions in the part that can be added to the design table.



Press **Ctrl** and select these four dimensions:

`CylinderDepth@Cylinder`

`AcrossFlats@Sketch2`

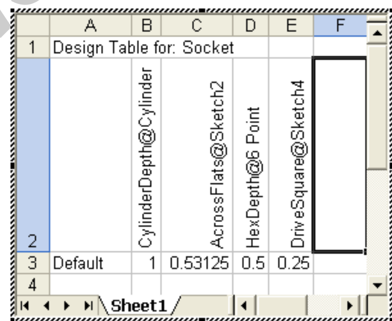
`HexDepth@6 Point`

`DriveSquare@Sketch4`

Tip The dimensions that will be used in the table should be renamed to more meaningful names than their default ones.

14 Design table.

The selected dimensions are added to the design table with their associated values. Note that the long dimension names are automatically rotated vertical.



Excel Formatting

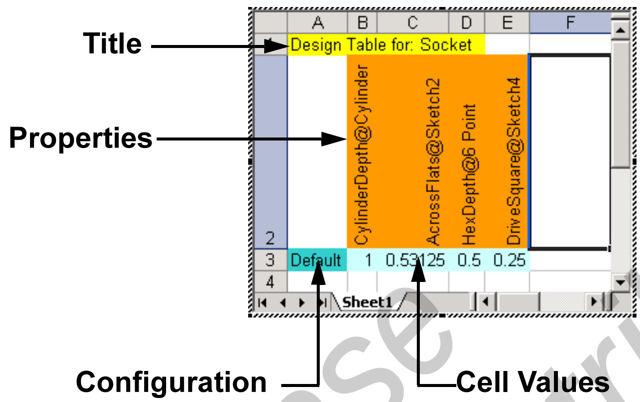
The cells in the spreadsheet appear with the standard formatting but they can be changed at any time. Take advantage of the power of Excel to make the design table easier to read and use. You can:

- Change cell colors and borders
- Change text color, orientation and font
- Define functions between cells

All of these changes can make the design table more useful or just more readable.

Anatomy of a Design Table

The design table contains rows and columns of information that are set into predefined cells of the Excel spreadsheet.



Properties Used in the Design Table

The **Properties** that live in Row **2** of the design table can be used to set a dimension value, suppress or unsuppress a feature or add a comment. The chart below summarizes the available properties and valid cell values.

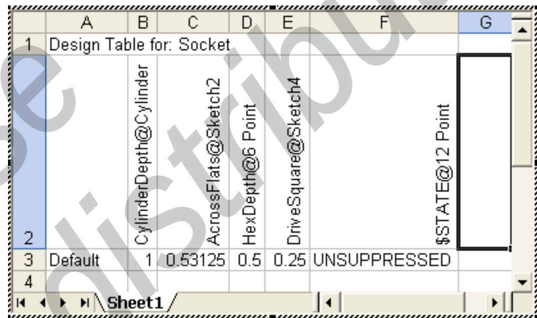
Property	Header Example	Cell Value	Description
Dimension	`D3@Sketch2`	Number	The value should be appropriate for the dimension.
Tolerance	`$TOLERANCE@D1@Sketch1`	Tolerance type (text); Maximum Variation (number); Minimum Variation (number)	The format sets the type and values separated by semi-colons (;) for a single dimension.
State	`$STATE@Fillet5`	S, U, Suppressed, Unsuppressed, or blank. Blank = Unsuppressed	Sets the state of a feature to be suppressed or unsuppressed.
Color	`$COLOR`	32-bit integer number.	Color value is derived from palette color or material.
Parent	`$PARENT`	Text	Parent configuration name.
Comment	`$COMMENT`	Text	Alpha-numeric.
User Notes*	`$USER_NOTES`	Text	Alpha-numeric. * Can be used as a column or row header.
Property	`$prp@prop_name`	Text	A custom property name `prop_name` created in the table or through **File, Properties**.

Adding New Headers

New property headers, feature or dimension based, can be added to the table by double-clicking. For a feature, double-click it in the FeatureManager or on the screen. The result is a $STATE property (see *Properties Used in the Design Table* on page 312). For a dimension, double-click it in the graphics area.

Note The next available cell in row 2 *must be* selected before double-clicking.

15 Add feature.
Double-click the `12 Point` feature on the FeatureManager. It is added to the design table with the prefix `$STATE@`. The current state, `UNSUPPRESSED`, is also added.



	A	B	C	D	E	F	G
1	Design Table for: Socket						
2		CylinderDepth@Cylinder	AcrossFlats@Sketch2	HexDepth@6 Point	DriveSquare@Sketch4	$STATE@12 Point	
3	Default	1	0.53125	0.5	0.25	UNSUPPRESSED	
4							

Sheet1

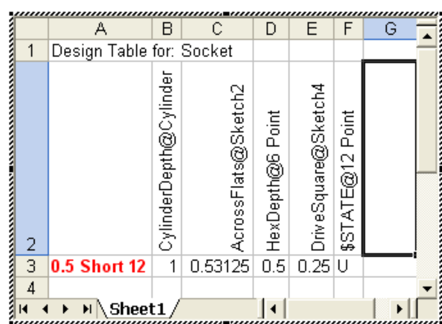
Change the value `UNSUPPRESSED` to the abbreviation `U`.

Note The `UNSUPPRESSED` cells can be abbreviated as **U**. `SUPPRESSED` cells can be abbreviated as **S**. A lowercase **u** or **s** would be converted to uppercase.

Adding Configurations to the Table

Once the design table has been established, configurations and associated cell values can be added by typing. In this example, a new configuration name will be created and copied to generate more.

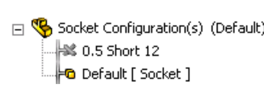
16 Add a new configuration.
Replace in the `Default` configuration name with `0.5 Short 12` in the cell `A3`. Close the table.



	A	B	C	D	E	F	G
1	Design Table for: Socket						
2		CylinderDepth@Cylinder	AcrossFlats@Sketch2	HexDepth@6 Point	DriveSquare@Sketch4	$STATE@12 Point	
3	0.5 Short 12	1	0.53125	0.5	0.25	U	
4							

Sheet1

17 Delete configuration.
Exit the design table confirming the creation of a new configuration.



In the Configuration Manager, make the new configuration active and delete `Default`.

18 Add more configurations.
Right-click the `Design Table` feature Design Table and select **Edit Table**. Add more configurations quickly by selecting cells `A3-F3` and dragging the lower right corner of the selection box down to row 5. Modify some of the cells as shown.

	A	B	C	D	E	F	G
1	Design Table for: Socket						
2		CylinderDepth@Cylinder	AcrossFlats@Sketch2	HexDepth@6 Point	DriveSquare@Sketch4	$STATE@12 Point	
3	0.5 Short 12	1	0.53125	0.5	0.25	U	
4	**0.625 Short 12**	1	**0.65625**	0.5	0.25	U	
5	**0.75 Short 12**	1	**0.78125**	0.5	0.25	U	
6							

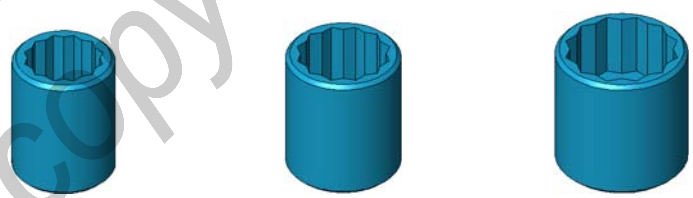
Sheet1

19 Configurations added.
Close the design table to add the new configurations. A message appears indicating the names of new configurations that have been added.

Tip

If configurations are missing from the list, there is a problem in the design table. Often the problem is an inappropriate cell value. If so, a message will appear, stating that a value is invalid.

20 Resulting configurations.
Switch to the Configuration Manager and double-click each configuration to make it active.



`0.5 Short 12` `0.625 Short 12` `0.75 Short 12`

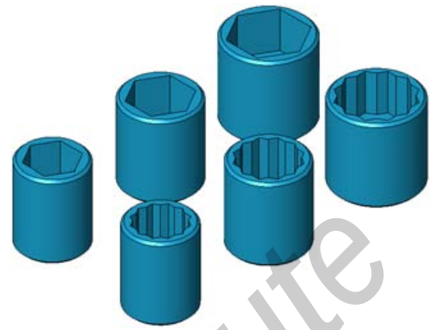
Only one configuration can be active at any time.

21 Add configurations and comments.
Using copy and paste with editing, create as many additional configurations, and formatting as time permits.

	A	B	C	D	E	F	G
1	Design Table for: Socket						
2		CylinderDepth@Cylinder	AcrossFlats@Sketch2	HexDepth@6 Point	DriveSquare@Sketch4	$STATE@12 Point	
3	$USER_NOTES						
4	0.5 Short 12	1	0.53125	0.5	0.25	U	
5	0.625 Short 12	1	0.65625	0.5	0.25	U	
6	0.75 Short 12	1	0.78125	0.5	0.25	U	
7	$USER_NOTES						
8	**0.5 Short 6**	1	0.53125	0.5	0.25	**S**	
9	**0.625 Short 6**	1	0.65625	0.5	0.25	**S**	
10	**0.75 Short 6**	1	0.78125	0.5	0.25	**S**	
11							

Sheet1

22 Configurations created.
If all configurations are created, there are 6 configurations. For each size there is a 6 and 12 point version.



Existing Design Tables

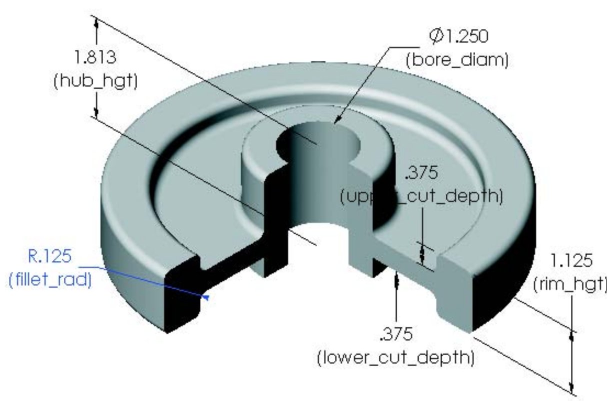
Another way to add a design table is to create the table in Excel and insert it into the part. Here are some tips when using this method:

- **Rename dimensions**
  As previously mentioned, the default dimension names are generally non-descriptive. Rename them by editing the **Properties** of the dimension and modifying the **Name** field.
- **Copy dimension and feature names**
  The design table is very fussy about the spelling and case of dimension and feature names. Use copy and paste to extract the **Full name** of the dimension from the **Properties** dialog and add it to the cell. For features, also use the **Properties** dialog.
- **Fill in all cells**
  All the cells within the rows and columns that you create must have the appropriate type of data in them.

1 Open the part named `Part_DT`.
The part, `Part_DT`, will be used to demonstrate the power of *existing* design tables.

The part has both revolved and extruded features. Note that multiple features are used where one revolved could be used. This allows the individual features to be suppressed.

2 Key dimensions.
Using properties, some key dimensions have been changed from their default names to something more descriptive. Only those that appear in the design table need to be changed.



If the default names are used, comments can be added to the design table to describe the dimension further.

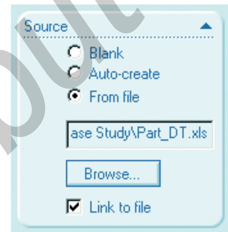
Tip

When it comes time to copy the dimension names into the spreadsheet, it will be easier to select the names if they do not have embedded spaces.

Inserting the Design Table

After creating the design table, it has to be inserted into the appropriate SolidWorks part. To do this, use the following procedure:

3 Insert the design table into the part.



Click **Design Table** or click **Insert, Design Table...**. For the **Source** select **From file**.

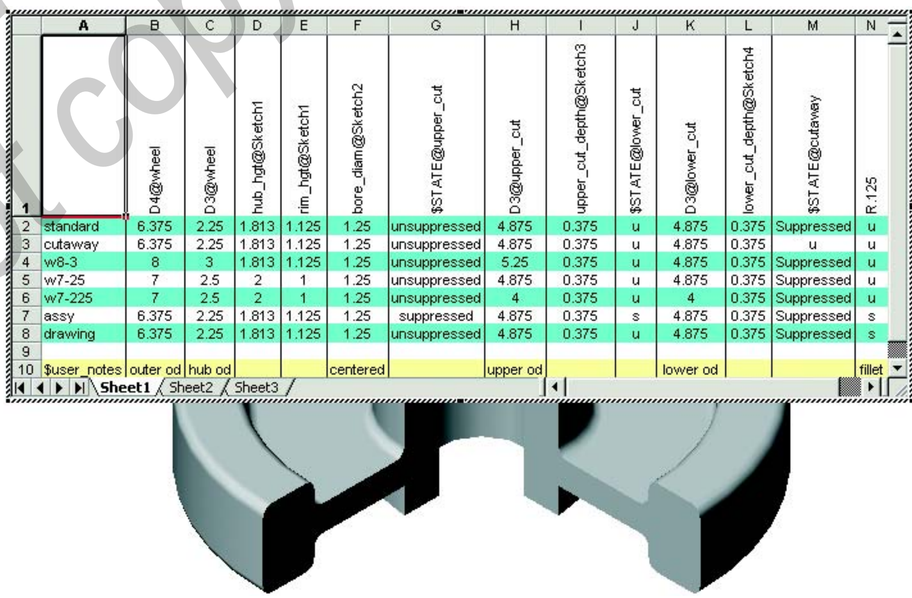
Select **Link to file** to tie the external file to the part. Click **Browse...**, and select the Excel file `Part_DT.xls` and click **OK**.

Note

If **Link to file** is *not* used, the spreadsheet will be copied into the part document and stored there.

4 Design table on screen.
The design table spreadsheet is linked into the part. Clicking outside the spreadsheet in the graphics window will close it.

In this example the existing design table has the same name as the part, `Part_DT`.

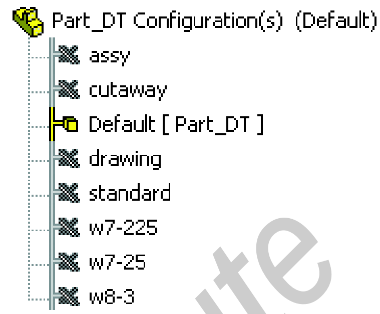


	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1		D4@wheel	D3@wheel	hub_hgt@Sketch1	rim_hgt@Sketch1	bore_diam@Sketch2	$STATE@upper_cut	D3@upper_cut	upper_cut_depth@Sketch3	$STATE@lower_cut	D3@lower_cut	lower_cut_depth@Sketch4	$STATE@cutaway	R.125
2	standard	6.375	2.25	1.813	1.125	1.25	unsuppressed	4.875	0.375	u	4.875	0.375	Suppressed	u
3	cutaway	6.375	2.25	1.813	1.125	1.25	unsuppressed	4.875	0.375	u	4.875	0.375	u	u
4	w8-3	8	3	1.813	1.125	1.25	unsuppressed	5.25	0.375	u	4.875	0.375	Suppressed	u
5	w7-25	7	2.5	2	1	1.25	unsuppressed	4.875	0.375	u	4.875	0.375	Suppressed	u
6	w7-225	7	2.5	2	1	1.25	unsuppressed	4	0.375	u	4	0.375	Suppressed	u
7	assy	6.375	2.25	1.813	1.125	1.25	suppressed	4.875	0.375	s	4.875	0.375	Suppressed	s
8	drawing	6.375	2.25	1.813	1.125	1.25	unsuppressed	4.875	0.375	u	4.875	0.375	Suppressed	s
9														
10	$user_notes	outer od	hub od			centered		upper od			lower od			fillet

5 Successful configurations.
A successful process will include a dialog that lists the configurations that were created.

Click **OK**.

6 Access the ConfigurationManager.
Access the ConfigurationManager and **Show Configuration...** for each of the new configurations.

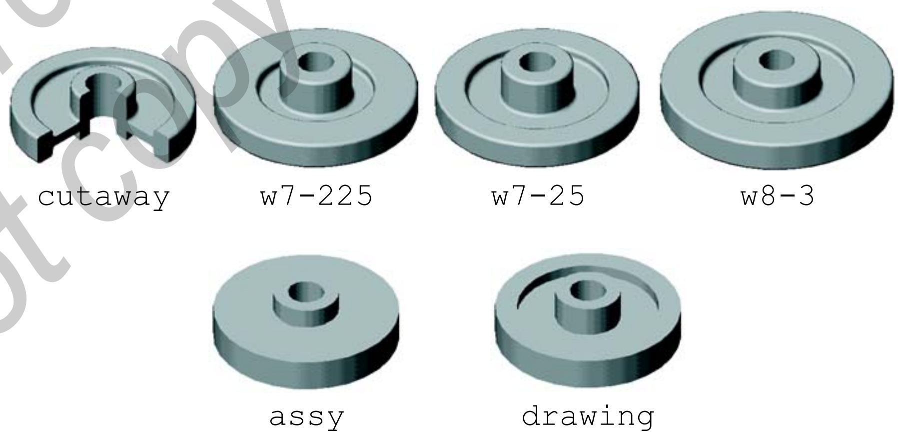


Note that the configurations are not listed in the order they were in the spreadsheet. They are listed alpha-numerically. Also note that the `Default` configuration has a different icon than the rest.

7 Deleting a configuration.
To delete a configuration, `Default` in this case, it must not be active. Click on the name and press the **Delete** key. Click **Yes** on the dialog to confirm deleting the configuration.

8 Save.
When the part is saved a message appears indicating that the design table is also being saved.

9 Configurations established.
Six configurations are established for the part. Each one is shown below. The `cutaway` configuration is the only one that does not suppress the `cutaway` feature.



Inserting Blank Design Tables

The **Blank** option is useful when configurations are needed for purposes other than feature suppression or setting of dimension values. Some examples would be:

- Only **Comments** (row or column) are required in the table.
- Multiple exploded views of an assembly.
- Multiple positions of a component in an assembly.
- Features and dimensions are to be added manually.

Tip

In general, if dimensions and features are to be controlled in the table, the **Auto-create** option is a better choice. The configurations can also be created outside a table, see *Defining the Configuration* on page 282.

Saving a Design Table

When you auto-create or insert a blank design table, the only place the table exists is embedded within the SolidWorks part. There may well be cases when you want to save the embedded table as an Excel spreadsheet.

Introducing: Save Table

Save Table allows you to save an embedded design table as an Excel spreadsheet.

Where to Find It

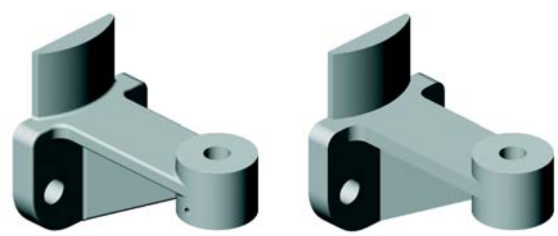
- Right-click the design table Design Table, and select **Save Table...**.

Other Uses of Configurations

Part configurations have numerous applications and uses. Some of the reasons for creating different configurations include:

- Application-specific requirements.
- Different product specifications such as a military and civilian version of a part.
- Performance considerations.
- Assembly considerations.

Application-specific Requirements



Many times the finished part model contains fine detail such as fillets and rounds. When preparing a part such as the one shown at the right for finite element analysis (FEA), it is desirable to simplify the part. By suppressing the unnecessary detail features you can create a configuration specifically for FEA.

Another application that might require a specialized model representation would be rapid prototyping.

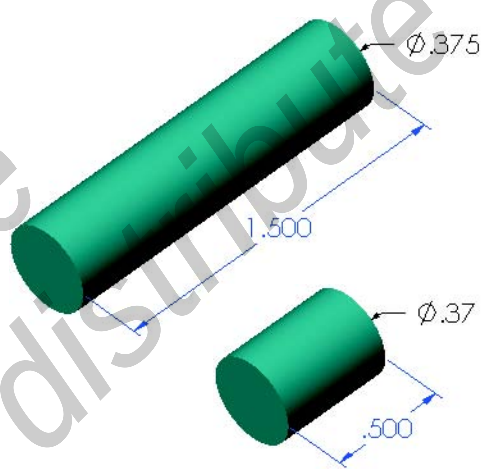
Performance Considerations

Parts with complex geometry such as swept and lofted features, variable radius fillets, and multi-thickness shells have a tendency to tax system resources. You might want to consider defining a configuration that suppresses some of these features. This will allow you to improve system performance when working on other, unrelated areas of the model. When you do this, however, be sure to take into account parent/child relations. You cannot access, use, or reference suppressed features – therefore they can't serve as parents.

Assembly Considerations

When working on complex assemblies that contain large numbers of parts, using simplified representations of those parts can improve system performance. Consider suppressing unnecessary detail such as fillets, leaving only critical geometry that is needed for mating, interference checking, and defining fit and function. When you add a component to an assembly, the **Insert, Component, From File...** browser allows you to choose the configuration of the part to be shown. To take best advantage of this, you have to plan ahead, defining and saving the configuration when the component is built.

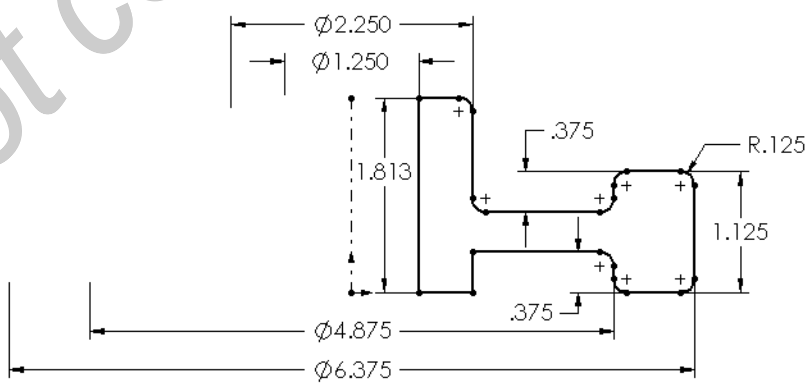
Similar parts that have the same basic shape can be defined as different configurations and used in the same assembly. The part shown at the right has two configurations. For an example showing how to use two different configurations of a part within an assembly see *Using Part Configurations in Assemblies* on page 396.



Modeling Strategies for Configurations

When you model a part that will be used with configurations – whether or not it is driven by a design table – you should give some thought to what you want the configurations to control. Consider, for example, the part used in the previous procedure.

One way a part like this can be modeled is to make a single sketch of the profile and build the part as a single revolved feature.

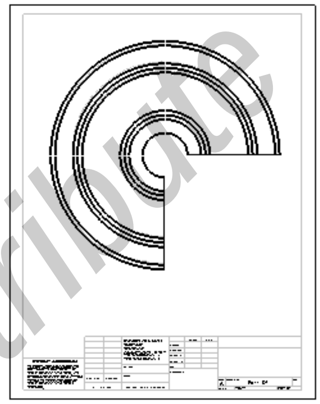


Although that approach seems efficient, having all the information contained in a single, monolithic feature really limits your flexibility. By breaking the part down into smaller, individual features, you gain the flexibility of being able to suppress features such as fillets or cuts.

More About Making Drawings

Drawings were first introduced in Lesson 3. In this section we will explore some additional detailing topics. These topics include: **Named Views**, **Section Views**, **Detail Views** and **Ordinate Dimensions**.

1 Open the drawing named `Part_DT`.



Drawing Properties

The drawing properties control many characteristics of each drawing sheet. Settings can be different for each drawing sheet.

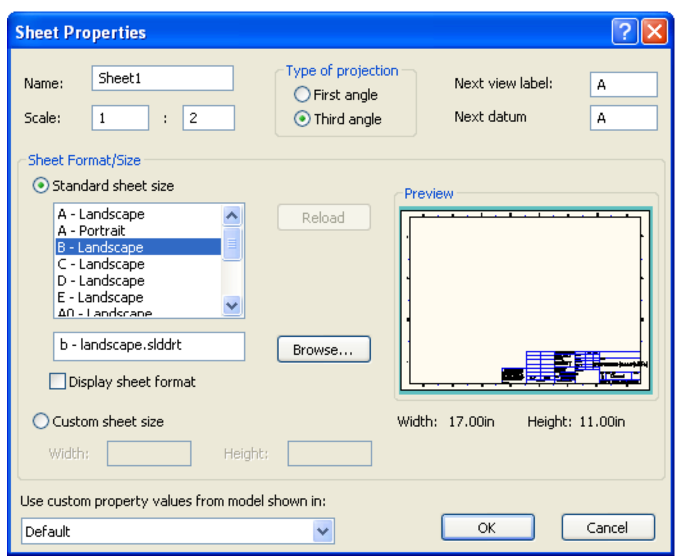
Introducing: Drawing Properties

View scale, paper size, sheet format and type of projection are all set or changed through the **Sheet Setup** dialog.

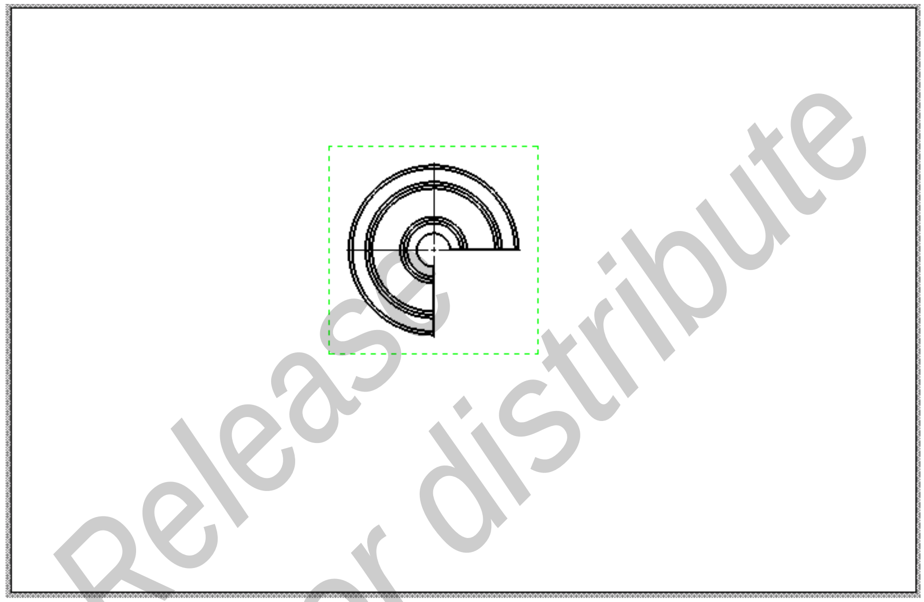
Where to Find It

- With the cursor over the drawing, right-click and select **Properties...**.
- Or, right-click the sheet icon Sheet1 in the FeatureManager, and select **Properties...**.

2 Sheet setup.
Right-click the sheet, and select **Properties...**. Set the **Standard sheet size** to `B-Landscape`, **Scale** to **1:2** and clear **Display sheet format**. Click **OK**.

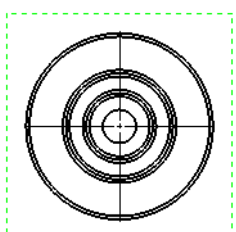


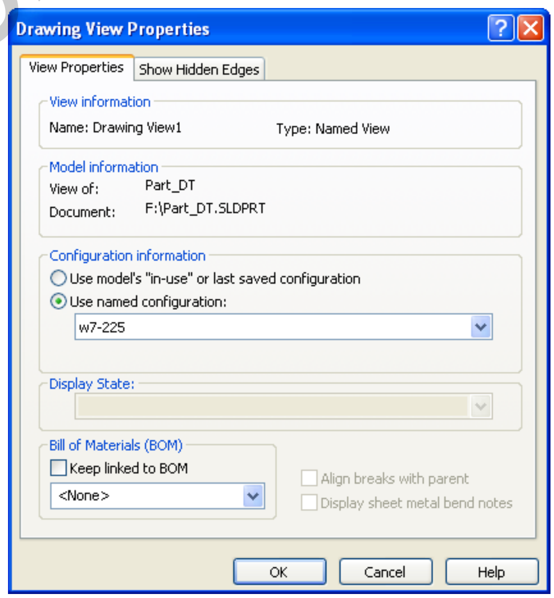
3 View scale.
Changes to the sheet view scale affect all views which do not use a custom scale.



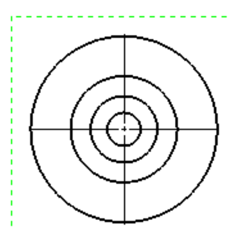
4 Change configuration.
Click in the drawing view and choose **More Properties...** from the **Model View** PropertyManager.

Click **Use named configuration** and select the configuration `w7-225`.





5 Tangent edges.
Right-click in the drawing view and select **Tangent Edge, Tangent Edges Removed**.



Simple Section View

You can create several types of section views. The *simple* section view uses a single line to form the cutting plane.

Introducing: Section View

Section View creates a full or partial section view based on a cutting line and a direction. A single sketch line is used for the section line.

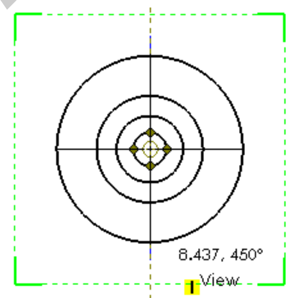
The fastest way to create the section is to click the tool first. This switches on the line tool for sketching the section line. When the line is completed, a preview of the section view appears.

Where to Find It

- From the menu click **Insert, Drawing View, Section**.
- Or, on the Drawing toolbar click the **Section View** tool.

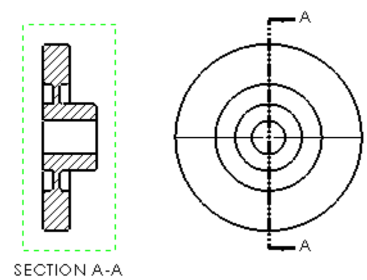
6 Click the Section View tool.

Click **Section View**. In the `Top` view, sketch a vertical line through the center of the part. Use the cursor feedback to align the line with the axis that runs through the center of the part. The line should extend well beyond the extents of the part.



7 Place the section view.

Move the cursor to the left of the view and place the section view by clicking the left mouse button.



The drawing view `Section A-A` is aligned to the source view and comes with a label beneath it. The cross hatching is automatic and reflects the type specified in the *part document* under **Tools, Options, System Options, Drawings, Area Hatch/Fill**.

Detail Views

Detail Views can be created using a closed sketched shape in an activated source view. The detail can use a scale multiplier to scale it *n* times larger than its source, default 2x. The contents of the detail view is determined by what is enclosed within the sketch. The sketch must be a closed contour but can be constructed of any sketch geometry types.

Introducing: Detail View

Detail View creates a new view of an area enclosed by a set of closed sketch geometry.

The fastest way to create the detail is to click the tool first. This switches on the circle tool for sketching the detail circle. When the circle is completed, the detail appears.

Where to Find It

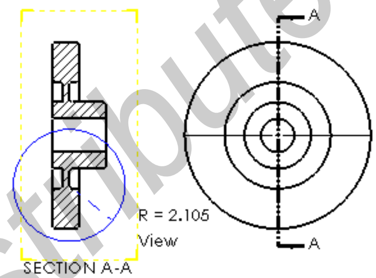
- From the menu click **Insert, Drawing View, Detail**.
- Or, on the Drawing toolbar click the tool.

---

8 Sketch the detail circle.

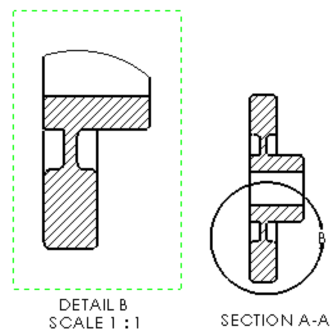
Click the **Detail** tool. In the `Section` view, sketch a circle as shown.

Use the cursor feedback to place the center of the circle on an endpoint of an edge. Drag the circle diameter to enclose the lower portion of the part.



9 Place the detail.

Position the view on the drawing by clicking the left mouse button. Set the detail view state to **Hidden Lines Removed**.



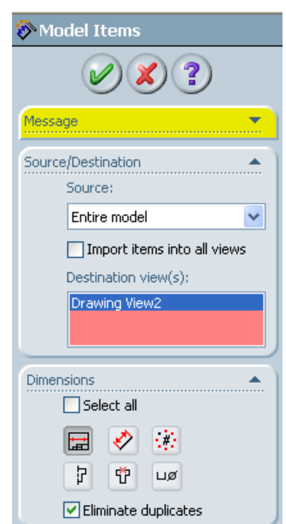
10 Insert Model Items.

Model items can be added to all views, selected views or selected features within a view.

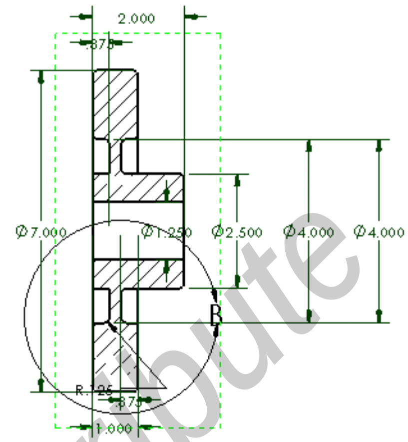
Select the section view, then click **Insert, Model Items...**.

Select **Marked for drawing**, **Import from: Entire model** and **Destination view(s)** (selected view).

Click **OK**.

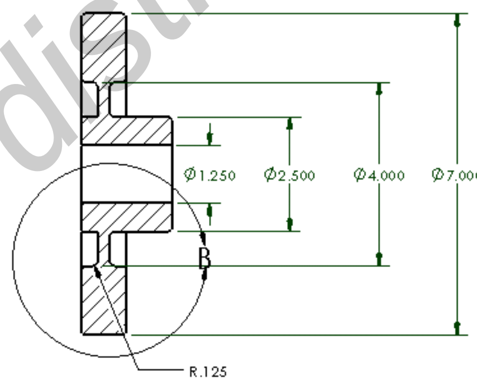


11 Dimensions in section view.
All the applicable dimensions are added to the section view.



12 Deleting dimensions.
Dimensions deleted at the drawing level are *not* deleted from the model.

Delete the dimensions for the depth of the part.



Annotations

Many annotation symbols can be added to a drawing. They include:

- **Datum Features**
- **Geometric Tolerance Symbols**
- **Notes**
- **Surface Finishes**
- **Weld Symbols**
- **Hole Callouts**
- **Balloons**
- **Cosmetic Threads**

Introducing: Datum Feature Symbol

The **Datum Feature Symbol** can be attached to model edges in the drawing views.

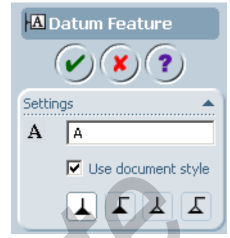
Where to Find It

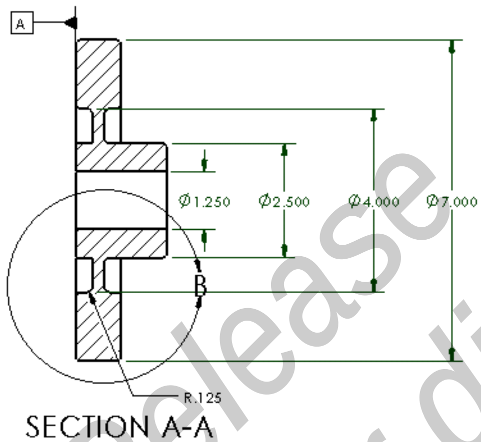
- On the Annotations toolbar, click **Datum Feature**.
- Or, click **Insert, Annotations, Datum Feature Symbol...**.

13 Add a Datum Feature Symbol.

Insert a **Datum Feature** with the **Label** A. Select the leftmost vertical model edge in the section view.

Click **OK**.





Ordinate Dimensions

In Lesson 3 you saw that you can manually add dimensions in the drawing document. These are reference dimensions, and are driven. That means you cannot edit the value of reference dimensions to change the model. However, changing a dimension in the model updates the driven dimension in the drawing.

SolidWorks supports several kinds of dimensions including **Ordinate Dimensions**. Ordinate dimensions are a set of dimensions measured from a zero ordinate in a drawing or sketch. They are measured from the axis you select first. The type of ordinate dimension (horizontal, vertical, or angular) is defined by the orientation of the points you select.

Ordinate dimensions are automatically grouped to maintain alignment. When you drag any member of the group, all the members move together. To disconnect a dimension from the alignment group, right-click the dimension, and select **Break Alignment**.

If adjacent dimensions are very close together, the leaders are automatically jogged as needed to prevent overlapping text. Drag handles are displayed at the bends when you select an ordinate dimension with a bent leader. You can remove the bend, or add a bend to a different ordinate dimension.

Where to Find It

- Click **Horizontal Ordinate Dimension** on the Dimensions/Relations toolbar.
- Click **Tools, Dimensions**, and **Horizontal Ordinate**.

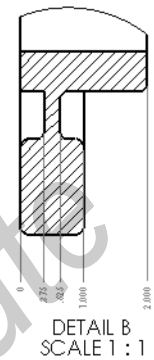
14 Add ordinate dimensions to the detail view.

Click **Horizontal Ordinate Dimension** on the Dimensions/Relations toolbar.

Select the leftmost edge of the part.

Click to position the **0** dimension.

Click the remaining edges you want to dimension. As you do so, each new ordinate dimension will automatically be placed on the drawing.



Parametric Notes

Using notes you can add text to a drawing. A note can be free floating or placed with a leader pointing to a face, edge, or vertex in the drawing. The note can contain simple text, symbols, parametric text, and hyperlinks. The leader can be straight or bent.

A parametric note is one that is linked to the value of a document property, a custom property, or a configuration specific property. If the value of the property changes, the note text changes automatically.

Where to Find It

- Click **Note** on the Annotations toolbar.
- Click **Insert, Annotations, Note...**.

15 Select the drawing view.

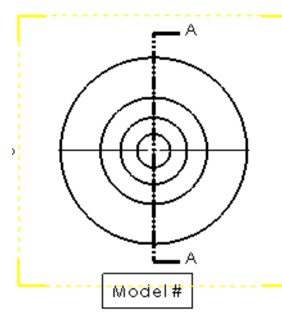
The note should be associated with the drawing view. To accomplish this, *double-click* the model view before you create the note.

16 Insert Note.

Click **Note** on the Annotations toolbar. In the PropertyManager, under **Leader**, click **No Leader** to create the text without a leader.

Click below the section line and type:
`Model #`

Be sure to type a space *after* the # character.

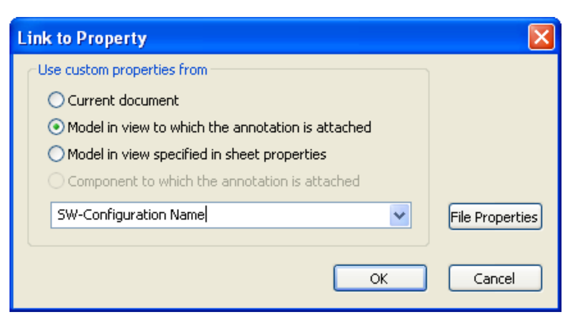


17 Link to property.

Click **Link to Property** on the **Note** PropertyManager.

Click **Model in view to which the annotation is attached**.

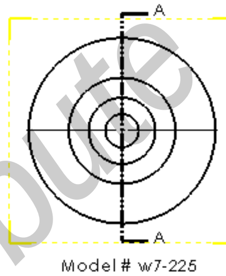
Select **SW-Configuration Name** from the list and click **OK**.



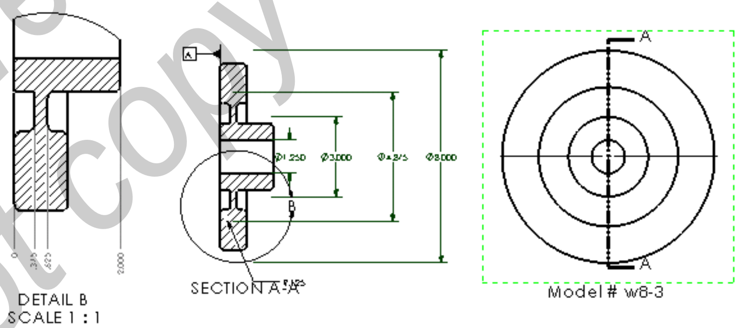
18 Font.
Select the font to be **11 point** and **Center**. Click **OK**.

19 Text of property.
The note shows the value of the property, the current configuration name, immediately.

This is a WYSIWYG display of the completed note.



20 Change the configuration.
Right-click `Drawing View1` and choose **Properties...**. In the **Model View** PropertyManager, click **More Properties...**. Under **Configuration information**, click **Use named configuration** and select `w8-3` from the list. All three views update to reflect the newly selected configuration. The dimensions also update to reflect the change in the model's size. The text of the note also updates, showing the current configuration name.



Introducing: Model View

Model Views are views which take their orientation and name from the **View Orientation** dialog in parts and assemblies. All standard views, user-defined views and the current view are eligible for use as a named view on a drawing sheet.

If the view selected in the model is a perspective view, that information is also carried into the drawing view

Where to Find It

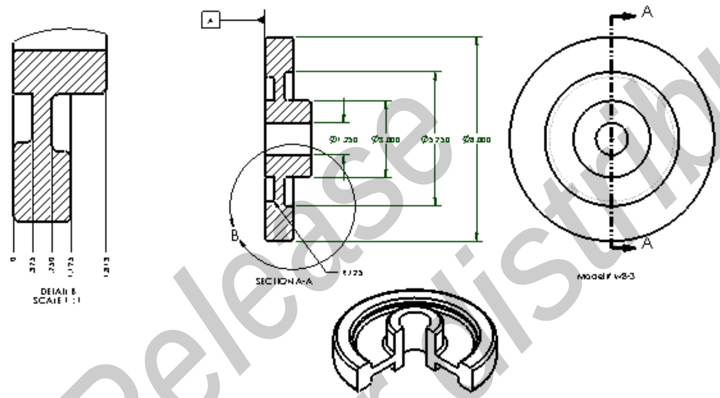
- Click **Model View** on the Drawing toolbar.
- Or, click **Insert, Drawing View, Model...**.

21 Add a Model View.

Click **Model View** on the Drawing toolbar.

To identify which model should appear in the view, click inside the `Top` view (`Drawing View1`).

Select `*Isometric` from the **View Orientation** and place the view on the drawing. Select `cutaway` as the configuration used in the view.



Area Hatch

When we created the section view, the software added the crosshatch automatically. You can also manually apply a crosshatch pattern to a solid face.

Where to Find It

- Click **Area Hatch/Fill** on the Drawing toolbar.
- Or, click **Insert, Annotations, Area Hatch/Fill**.

Note

This is one of the few commands in SolidWorks that requires you to preselect the geometry. You must first select the face. Otherwise the command remains unavailable.

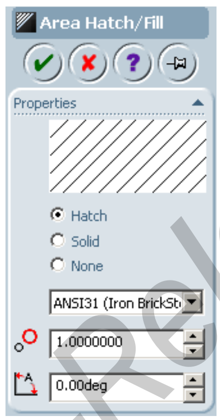
22 Area Hatch.

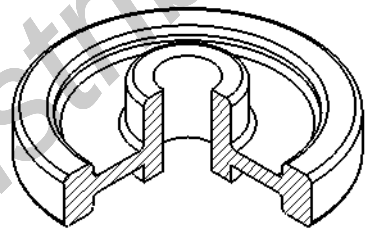
Select the two cut faces of the model in the isometric view.

Click **Area Hatch/Fill** .

The **Area Hatch/Fill** dialog box appears. This allows you to change the style of the hatch pattern.

Click **OK**.

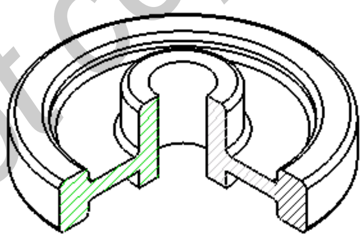


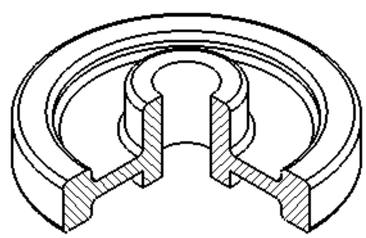


23 Edit the hatch pattern.

Click the leftmost section of area hatch. Change the **Angle** to **90°** and click **OK**.

This gives a more pleasing appearance to the hatch pattern.





Design Tables in a Drawing

The design table of a part can be shown on a drawing sheet. After selecting a view of the part, click **Insert, Tables, Design Table...** and place it on the drawing. Double-clicking the table opens the referenced part and the design table within it.

	D4@wheel	D3@wheel	hub_hgt@Sketch1	rim_hgt@Sketch1	bore_diam@Sketch2	$STATE@upper_cut	D3@upper_cut	upper_cut_depth@Sketch3	$STATE@lower_cut	D3@lower_cut	lower_cut_depth@Sketch4	$STATE@cutaway	R.125	$STATE@R.125
standard	6.375	2.25	1.813	1.125	1.25	U	4.875	0.375	U	4.875	0.375	S	u	U
cutaway	6.375	2.25	1.813	1.125	1.25	U	4.875	0.375	U	4.875	0.375	U	u	U
w8-3	8	3	1.813	1.125	1.25	U	5.25	0.375	U	4.875	0.375	S	u	U
w7-25	7	2.5	2	1	1.25	U	4.875	0.375	U	4.875	0.375	S	u	U
w7-225	7	2.5	2	1	1.25	U	4	0.375	U	4	0.375	S	u	U
assy	6.375	2.25	1.813	1.125	1.25	S	4.875	0.375	S	4.875	0.375	S	s	S
drawing	6.375	2.25	1.813	1.125	1.25	U	4.875	0.375	U	4.875	0.375	S	s	S
$user_note	outer od	hub od			centered		upper od			lower od			fillet	

24 Insert the design table.
Click inside one of the drawing views. Since a drawing can contain views of several different models, you have to identify from which part the design table will be inserted.

Click **Design Table** or click **Insert, Tables, Design Table...**.

The design table appears in the upper left corner of the drawing. Drag it to where you want it on the drawing.

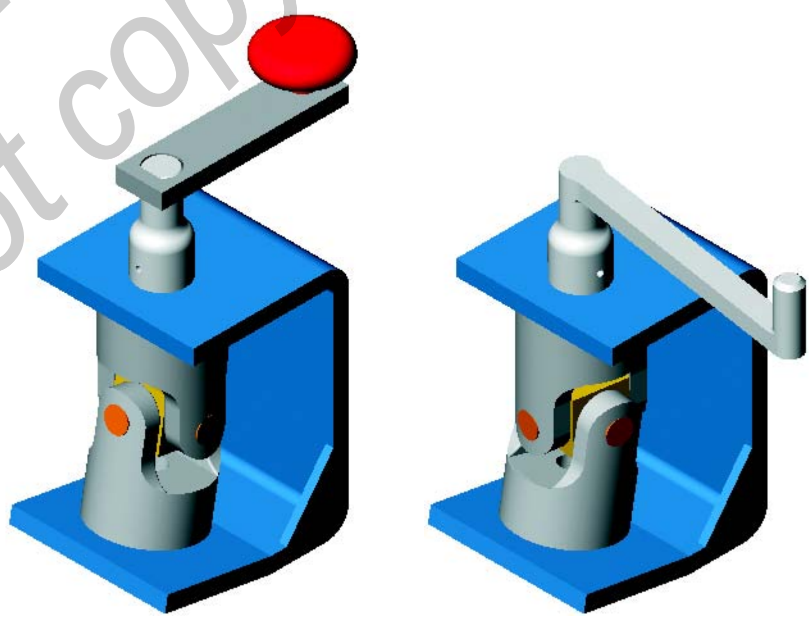
For more information about making drawings in SolidWorks, you should attend the *SolidWorks Essentials: Drawings* course.

In the Advanced Course...

In the advanced course *Advanced Assembly Modeling*, the concept of **Configurations** is carried into assemblies.

Assemblies can have configurations that are created manually or through a design table. While part configurations focus on features, assembly configurations focus on components, mates, or assembly features. Assembly configurations can be used to control:

- Assembly Features
- Components
- Mates and Mate Dimensions



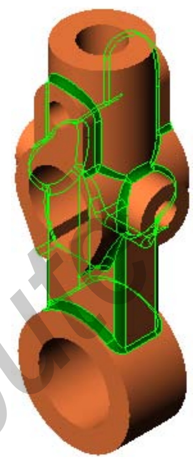
Design tables can also be used. At the assembly level there are more options available to control one or more component instances.

Exercise 40: Using Link Values

Create link values in an existing part and test it.

This lab reinforces the following skills:

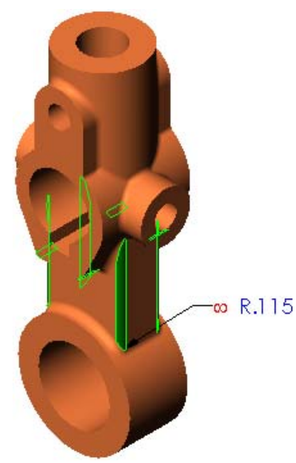
- Creating link values.



Procedure

Open the existing part named `Link Values`. Create a link value that makes all the fillets feature values equal.

1. **Create link value.**
   Create and apply a link value named `All_fillets&rounds` to the dimension of the `Rounds` feature.



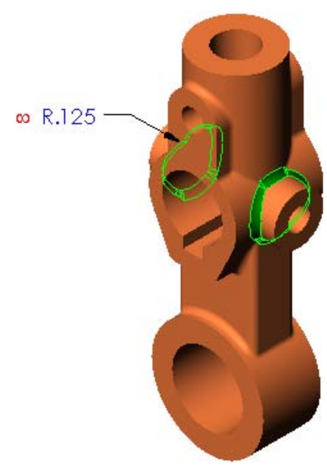
2. **Apply link value.**
   Apply the link value to the remaining three fillet features:

   `Fillets.1`

   `Fillets.2`

   `Fillets.3`

3. **Test.**
   Test the links by changing any one of the four to **0.125"** and rebuilding.



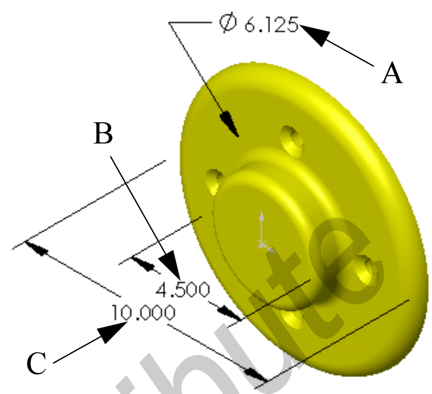
4. **Save and close the part.**

Exercise 41: Using Equations

Create an equation using an existing part and test it.

This lab reinforces the following skills:

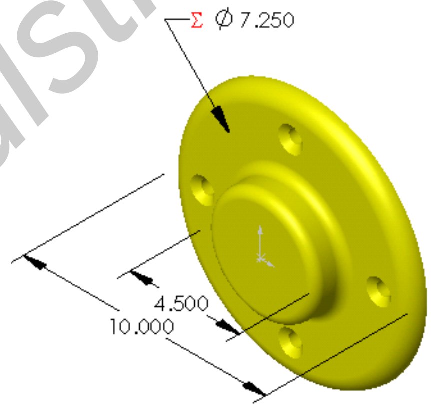
- Creating equations.



Procedure

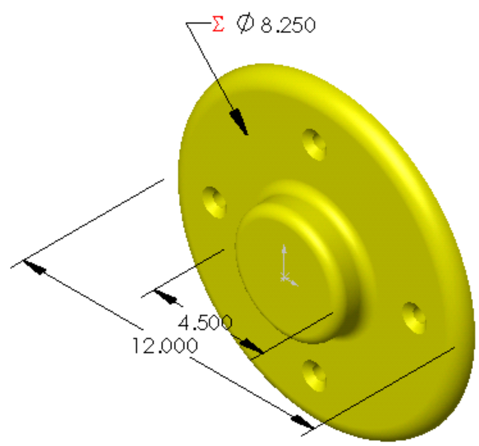
Open the existing part named `Using Equations`. The dimensions **A**, **B** and **C** shown above will be used to define the equation.

1 **Write equation.**
Write an equation that keeps the bolt circle diameter (**A**) centered between the outside edges of the hub (**B**) and flange (**C**). The (**A**) value should be *driven*.



2 **Test equation.**
Test the equation be changing the flange diameter to **12”** and rebuilding the model. Test other values if you wish.

3 **Save and close the part.**

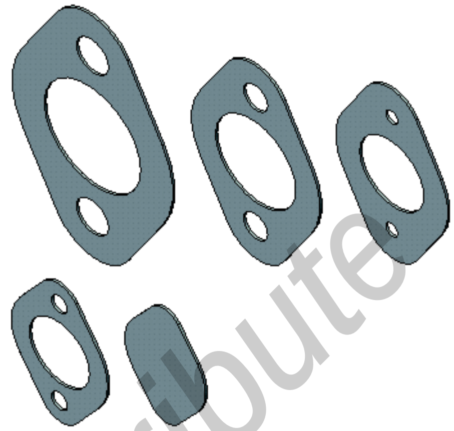


Tip

If you are having trouble, the equation format should be: `A=(C-B)/2+B`.

Exercise 42: Part Design Tables

Use an existing part as the basis for a design table. Use the dimensions and add them into a new design table.



This lab reinforces the following skills:

- Inserting design tables.
- Editing design tables.
- Adding properties.
- Using configurations.

Procedure

Open the existing part `Part Design Table`.

1 Design table.
Create a design table using **Auto-create** and edit it as shown.

2 Add dimensions.
CTRL-select all the dimensions, with the exception of `D1@Main`, in the **Dimensions** dialog.

Add them to the design table. The current values are added automatically.

	A	B	C	D	E	F	G
1	Design Table for: Part Design Table						
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	
3	default	1.25	2.5	5	1	3	
4							

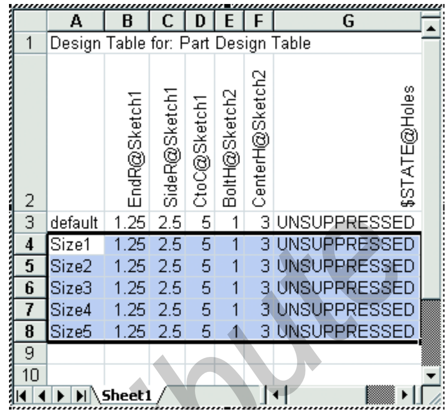
3 Add feature.
Double-click the `Holes` feature to add it to the design table. The current state is added automatically.

	A	B	C	D	E	F	G
1	Design Table for: Part Design Table						
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes
3	default	1.25	2.5	5	1	3	UNSUPPRESSED
4							

4 Add configuration.
Type in the configuration name `Size1` as shown. Copy the cells as shown.

	A	B	C	D	E	F	G
1	Design Table for: Part Design Table						
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes
3	default	1.25	2.5	5	1	3	UNSUPPRESSED
4	Size1	1.25	2.5	5	1	3	UNSUPPRESSED
5							
6							
7							
8							
9							
10							

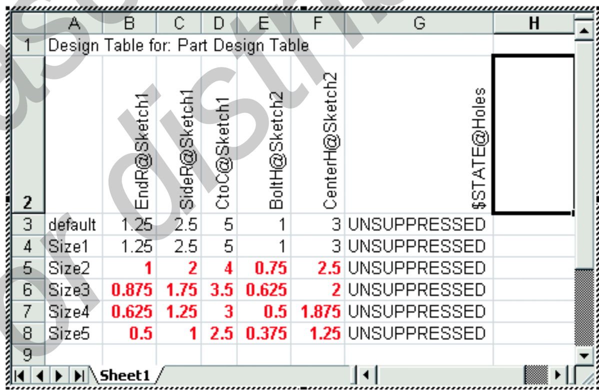
Copy the row, including the configuration, to add additional configurations.



	A	B	C	D	E	F	G
1	Design Table for: Part Design Table						
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes
3	default	1.25	2.5	5	1	3	UNSUPPRESSED
4	Size1	1.25	2.5	5	1	3	UNSUPPRESSED
5	Size2	1.25	2.5	5	1	3	UNSUPPRESSED
6	Size3	1.25	2.5	5	1	3	UNSUPPRESSED
7	Size4	1.25	2.5	5	1	3	UNSUPPRESSED
8	Size5	1.25	2.5	5	1	3	UNSUPPRESSED
9							
10							

Sheet1

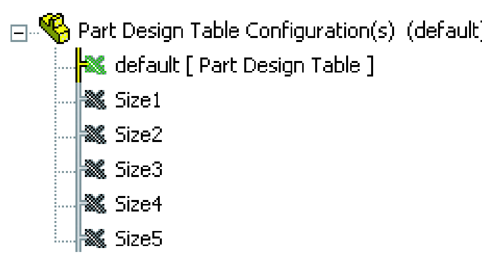
5 Edit cells.
Edit the cells for the `Size2` to `Size5` configurations. The changes are shown in bold red text.



	A	B	C	D	E	F	G	H
1	Design Table for: Part Design Table							
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes	
3	default	1.25	2.5	5	1	3	UNSUPPRESSED	
4	Size1	1.25	2.5	5	1	3	UNSUPPRESSED	
5	Size2	**1**	**2**	**4**	**0.75**	**2.5**	UNSUPPRESSED	
6	Size3	**0.875**	**1.75**	**3.5**	**0.625**	**2**	UNSUPPRESSED	
7	Size4	**0.625**	**1.25**	**3**	**0.5**	**1.875**	UNSUPPRESSED	
8	Size5	**0.5**	**1**	**2.5**	**0.375**	**1.25**	UNSUPPRESSED	
9								

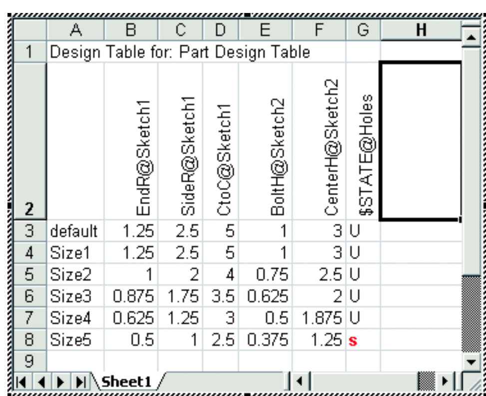
Sheet1

6 Close the design table.
Click outside the design table to close it. It should create five new configurations. The names are the same names that appear in column `A` of the spreadsheet.



7 Try the configurations.
Choose each of the configurations from the ConfigurationManager and test them.

8 Edit the design table.
Edit the design table using **Edit Table...**. Set the state to suppressed for the `Holes` feature in configuration `Size5`. Click outside the design table to apply the changes.



	A	B	C	D	E	F	G	H
1	Design Table for: Part Design Table							
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes	
3	default	1.25	2.5	5	1	3	U	
4	Size1	1.25	2.5	5	1	3	U	
5	Size2	1	2	4	0.75	2.5	U	
6	Size3	0.875	1.75	3.5	0.625	2	U	
7	Size4	0.625	1.25	3	0.5	1.875	U	
8	Size5	0.5	1	2.5	0.375	1.25	**s**	
9								

Sheet1

9 Test the edited configuration.
Test the configurations, focusing on `Size5`. The feature `Holes` should be suppressed in that configuration.

10 (Optional) Add spreadsheet functions.

Edit the **Design Table** to establish relationships between cells in the spreadsheet. Make the Side Radius (`SideR`) equal to half the center to center (`CtoC`) distance. For example, cell `C3` will be = `D3/2`; cell `C4` will be = `D4/2` and so on.

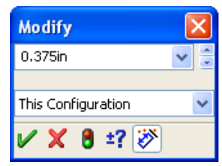
A `USER_NOTE` entry can be added to explain the relationship between the columns.

	A	B	C	D	E	F	G	H
1	Design Table for: Part Design Table							
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes	
3	default	1.25	2.5	5	1	3	U	
4	Size 1	1.25	2.5	5	1	3	U	
5	Size 2	1	2	4	0.75	2.5	U	
6	Size 3	0.875	1.75	3.5	0.625	2	U	
7	Size 4	0.625	1.5	3	0.5	1.875	U	
8	Size 5	0.5	1.25	2.5	0.375	1.25	S	
9	$USER_NOTES	SideR=CtoC/2						
10								

Sheet1

11 (Optional) Changes.

Make the `Size3` configuration active. Double-click the `Holes` feature and change the value of the `BoltH` dimension to **0.375”** for **This Configuration**. Click **OK** on the message box.



12 Bi-directional changes.

The change is made to the active configuration for that dimension. The change to the model forces a change in the design table.

	A	B	C	D	E	F	G	H
1	Design Table for: Part Design Table							
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes	
3	default	1.25	2.5	5	1	3	U	
4	Size 1	1.25	2.5	5	1	3	U	
5	Size 2	1	2	4	0.75	2.5	U	
6	Size 3	0.875	1.75	3.5	0.375	2	U	
7	Size 4	0.625	1.5	3	0.5	1.875	U	
8	Size 5	0.5	1.25	2.5	0.375	1.25	S	
9	$USER_NOTES	SideR=CtoC/2						
10								

Sheet1

13 Save and close the part.

Exercise 43: Existing Configurations and Linked Design Tables

Use existing parts to automatically create design tables and link to external Excel spreadsheets.

This lab reinforces the following skills:

- Automatically creating design tables from existing configurations.
- Inserting design tables.
- Linking to external design tables.

Auto-create

Open the existing part named `Auto-Create`. It contains several configurations but no design table.

1 Auto-create.
Insert a design table using the **Auto-create** option.

2 Design table.
A design table is generated from the existing configurations.

3 Save and close the part.

	A	B	C	D	E	F	G
1	Design Table for: Auto-Create						
2		EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	BoltH@Sketch2	CenterH@Sketch2	$STATE@Holes
3	default	0.625	1.5	3	0.5	1.875	
4	G1	1.25	2.25	4.5	1	3	
5	G2	1	2	4	0.75	2.5	
6	G3	0.875	1.75	3.5	0.625	2	
7	G4	0.625	1.5	3	0.5	1.875	
8	G5	0.5	1.25	2.5	0.375	1.25	S

Sheet1

Link to External Excel Spreadsheet

Open the existing part `Linked`. It contains no configurations except the `Default`.

4 From file.
Insert a design table using the **From file** option. Select the file `Design Table.xls`. Select the option **Link to file**.

5 Edit Excel file.
Open the linked Excel file `Design Table.xls` and add a new configuration `G6` as shown.

6 Return to the part.
Save and close the spreadsheet. Return to the part to see the updates.

7 Save and close the part.

	A	B	C	D	E	F	G	H
1		$PARTNUMBER	BoltH@Sketch2	CenterH@Sketch2	EndR@Sketch1	SideR@Sketch1	CtoC@Sketch1	$STATE@Holes
2	default	$D	0.5	1.875	0.625	1.5	3	U
3	G1	$D	1	3	1.25	2.25	4.5	U
4	G2	$D	0.75	2.5	1	2	4	U
5	G3	$D	0.625	2	0.875	1.75	3.5	U
6	G4	$D	0.5	1.875	0.625	1.5	3	U
7	G5	$D	0.375	1.25	0.5	1.25	2.5	S

Sheet1 Sheet2 S

Exercise 44: Designing for Configurations

Create a new part and design table. Design the part with the use of configurations and design tables in mind.

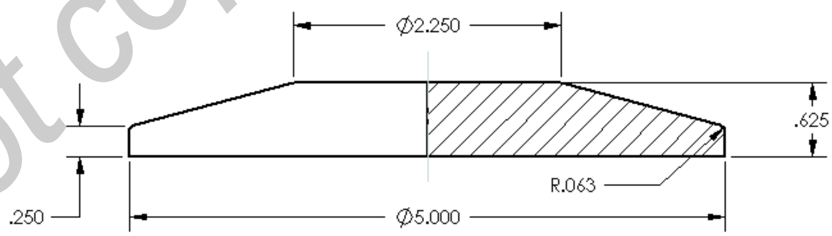
This lab reinforces the following skills:

- Modeling for configurations.
- Creating design tables in Excel or within SolidWorks.
- Using configurations.
- Excel options.

Procedure

Create a new part using the `Part_IN` template. Name the part `Design for Configs`.

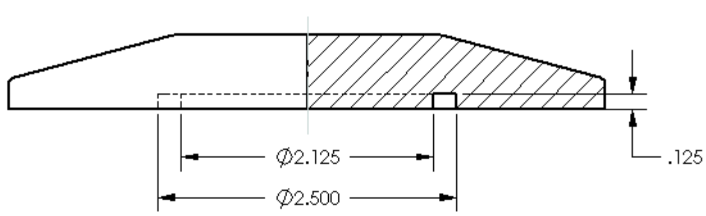
1 `Default` configuration.
Create the basic shape of the part as a revolved feature using the dimensions shown.



2 Dimension names.
Rename the overall diameter (5”) dimension to `Main_OD`.

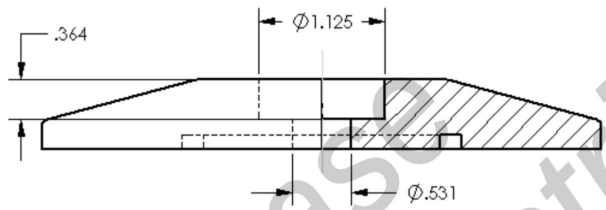
3 Groove Feature.
Create a cut feature to represent the groove. Name the feature `Groove`.

Rename the dimensions of the groove to `Groove_ID`, `Groove_OD` and `Groove_Depth`.



4 Counterbored hole.
Use the **Hole Wizard** to create a through all CBORE hole for a 1/2" Hex Head bolt.

The Hole Wizard generates dimensions with descriptive names. The names `C'Bore Dia.@Sketch3`, `C'Bore Depth@Sketch3`, `Thru Hole Depth@Sketch3` and `Thru Hole Dia.@Sketch3` are generated automatically.



5 Design table.
Use **Insert, Design Table** with the **Auto-create** option to create a design table within the part. Use automated and double-click methods to create the table shown below. Rearrange columns if necessary.

Design Table for: Design for Configs

	Main_OD@Sketch1	$STATE@Groove	Groove_Depth@Sketch3	Groove_ID@Sketch3	Groove_OD@Sketch3	$STATE@CBORE for 1/2 Hex Head Bolt1	C'Bore Dia.@Sketch4	Thru Hole Dia.@Sketch4	C'Bore Depth@Sketch4
Default	5	U	0.125	2.125	2.5	U	1.125	0.5312	0.364

Sheet1

6 Edit the design table.
Continue editing the design table, modifying it to include the values and additional configurations shown below. Add three more `Groove` configurations, suppressing all the `Cbore` features.

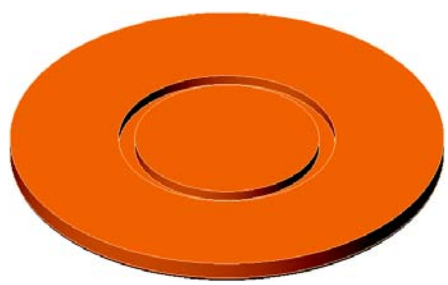
Design Table for: Design for Configs

	Main_OD@Sketch1	$STATE@Groove	Groove_Depth@Sketch3	Groove_ID@Sketch3	Groove_OD@Sketch3	$STATE@CBORE for 1/2 Hex Head Bolt1	C'Bore Dia.@Sketch4	Thru Hole Dia.@Sketch4	C'Bore Depth@Sketch4
Groove1	5	U	0.125	2.125	2.5	S	1.125	0.5312	0.364
Groove2	5.25	U	0.125	2.25	2.625	S	1.125	0.5312	0.364
Groove3	5.375	U	0.125	2.375	2.75	S	1.125	0.5312	0.364
Groove4	5.5	U	0.1875	2.5	2.875	S	1.125	0.5312	0.364

Sheet1

Note that the `Default` configuration has been replaced.

7 Test the configurations.
Check the four `Groove` configurations to see if they work properly. While one of the `Groove` configurations is active, delete the `Default` configuration.



Note You can add comments and other data to the spreadsheet by simply leaving a blank column between the design table data and the comments. You can also color blocks of cells to make it easier to recognize and associate groups of data.

8 Add more configurations.

Edit the design table again and add four more configurations for the `Cbore`. Yellow indicates copied information, bold red changed.

	A	B	C	D	E	F	G	H	I	J	K	L
1	Design Table for: Design for Configs											
2		Main_OD@Sketch1	$STATE@Groove	Groove_Depth@Sketch3	Groove_ID@Sketch3	Groove_OD@Sketch3	$STATE@CBORE for 1/2 Hex Head Bolt1	C'Bore Dia.@Sketch4	Thru Hole Dia.@Sketch4	C'Bore Depth@Sketch4		
3	Groove1	5	U	0.125	2.125	2.5	S	1.125	0.5312	0.364		
4	Groove2	5.25	U	0.125	2.25	2.625	S	1.125	0.5312	0.364		
5	Groove3	5.375	U	0.125	2.375	2.75	S	1.125	0.5312	0.364		
6	Groove4	5.5	U	0.1875	2.5	2.875	S	1.125	0.5312	0.364		
7	**Cbore1**	5	**S**	0.125	2.125	2.5	**U**	1.125	0.5312	0.364		1/2' Hex Bolt
8	**Cbore2**	5.25	**S**	0.125	2.25	2.625	**U**	**1.209**	**0.709**	**0.423**		M16 Hex Bolt
9	**Cbore3**	5.375	**S**	0.125	2.375	2.75	**U**	**1.482**	**0.866**	**0.528**		M20 Hex Bolt
10	**Cbore4**	5.5	**S**	0.1875	2.5	2.875	**U**	**1.482**	**0.866**	**0.528**		M20 Hex Bolt
11												

Sheet1

9 Save the part.

Exercise 45: Drawings

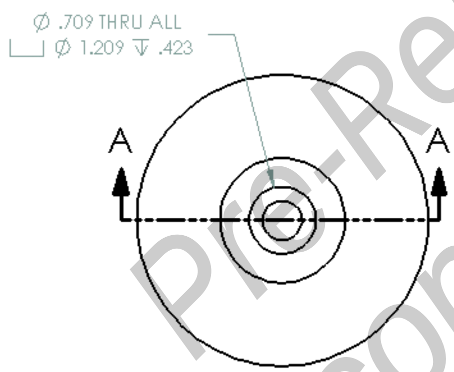
Create a drawing of the part you built in the previous exercise.

This lab reinforces the following skills:

- Creating named views.
- Creating a section view.
- Showing different configurations of a part in drawing views.
- Creating a parametric note.
- Adding dimensions and annotations.
- Adding hole callouts.
- Inserting a design table into a drawing.

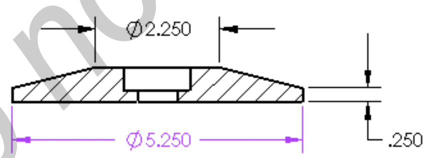
Procedure

Create an A-size drawing similar to the one shown below.

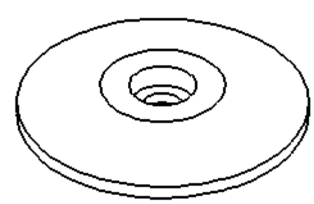


Design Table for: Design for Configs

	Main_OD@Sketch1	$STATE@Groove	Groove_Depth@Sketch3	Groove_ID@Sketch3	Groove_OD@Sketch3	$STATE@CBORE for 1/2 Hex Head Bolt1	C'Bore Dia.@Sketch4	Thru Hole Dia.@Sketch4	C'Bore Depth@Sketch4	
Groove1	5	U	0.125	2.125	2.5	S	1.125	0.5312	0.364	
Groove2	5.25	U	0.125	2.25	2.625	S	1.125	0.5312	0.364	
Groove3	5.375	U	0.125	2.375	2.75	S	1.125	0.5312	0.364	
Groove4	5.5	U	0.1875	2.5	2.875	S	1.125	0.5312	0.364	
Cbore1	5	S	0.125	2.125	2.5	U	1.125	0.5312	0.364	1/2' Hex Bolt
Cbore2	5.25	S	0.125	2.25	2.625	U	1.209	0.709	0.423	M16 Hex Bolt
Cbore3	5.375	S	0.125	2.375	2.75	U	1.482	0.866	0.528	M20 Hex Bolt
Cbore4	5.5	S	0.1875	2.5	2.875	U	1.482	0.866	0.528	M20 Hex Bolt



SECTION A



Cbore2

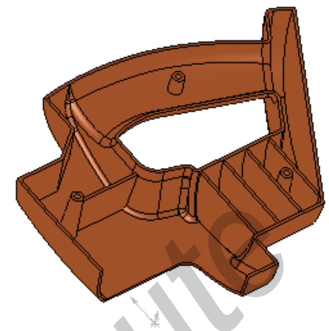
Lesson 11
Shelling and Ribs

Upon successful completion of this lesson, you will be able to:

- Apply draft to model faces.
- Use the rollback bar.
- Perform shelling operations to hollow out a part.
- Create reference planes.
- Use the rib tool.
- Create thin features.

Shelling and Ribs

Creating thin walled parts involves some common sequences and operations, whether they are cast or injection molded. Both shelling and draft are used, as well as ribs. This example will go through the steps of adding draft, creating planes, shelling and creating ribs.



Stages in the Process

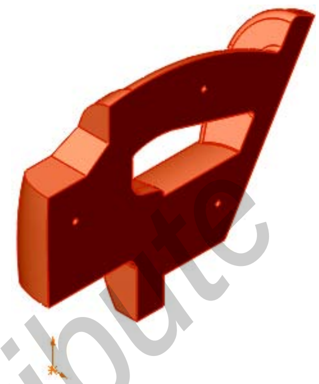
Some key stages in the modeling process of this part are given in the following list:

- **Draft with a reference plane**
  Draft can be defined with respect to a reference plane and direction.
- **Using planes**
  This part contains several features that are aligned to the centerline of the part itself. A centered plane is used for locating features.
- **Shelling**
  Shelling is the process of hollowing out a part. You have the option of removing one or more faces of the part. A shell feature is a type of applied feature.
- **Rib tool**
  The rib tool can be used to quickly create single or multiple ribs. Using minimal sketch geometry, the rib is created between bounding faces of the model.
- **Thin features**
  The thin feature option creates revolves, extrusions, sweeps and lofts with thin walls of constant thickness.

Analyzing and Adding Draft

Draft is required for both cast and injection molded parts. Because draft can be created in several ways, it is important to be able to check the draft on a part and if necessary, add more.

1 Open the part `Shelling&Ribs`.



Draft Analysis

The **Draft Analysis** tool is useful in determining whether the part has sufficient draft to be removed from the mold based on a set draft angle.

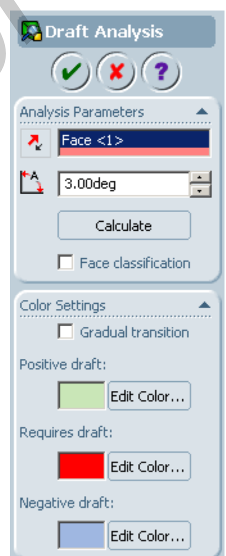
Where to Find It

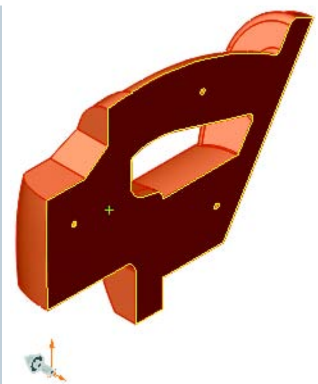
- Click **Draft Analysis** on the Mold Tools toolbar.
- Or, click **Tools, Draft Analysis...**.

2 Click Tools, Draft Analysis.

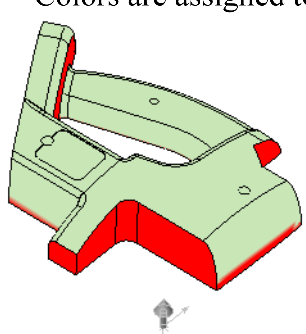
3 Direction of pull.
Select the rear planar face as the **Direction of Pull**. Click **Reverse Direction** so the pull direction arrow points as shown.

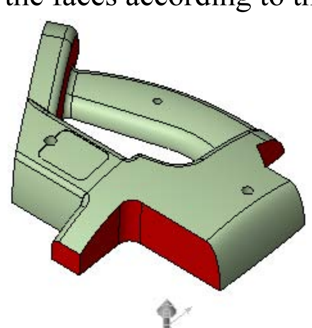
Set the **Angle** to **3°** and click **Calculate**.

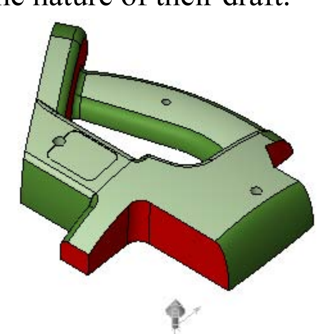




4 Results.
Colors are assigned to the faces according to the nature of their draft.


Require Draft

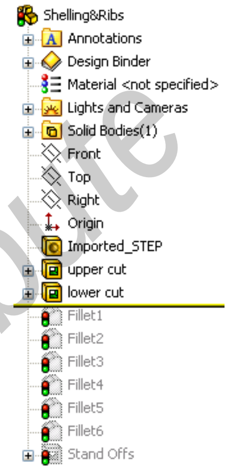

Face classification


Find steep faces

The **Face classification** and **Find steep faces** options produce more specific results. Click **OK** to complete the command. The draft is insufficient.

5 **Rollback.**
Draft must be added at an earlier stage of modeling. Rollback to a position between the `lower cut` and `Fillet1` features.

See *Introducing: The Rollback Bar* on page 235 for information on the rollback bar.



Other Options for Draft

So far we have seen one method for creating features with draft:

- Using the **Draft** option in the **Insert, Boss/Base, Extrude...** command.

There are times when this method does not address your specific situation. For example, because of the way we modeled the first feature, there isn't any draft on it. Clearly, there has to be a way to add draft to faces *after* they are created.

Introducing: Insert Draft

Insert Draft enables you to add draft to faces of the model with respect to a neutral plane or a parting line.

Where to Find It

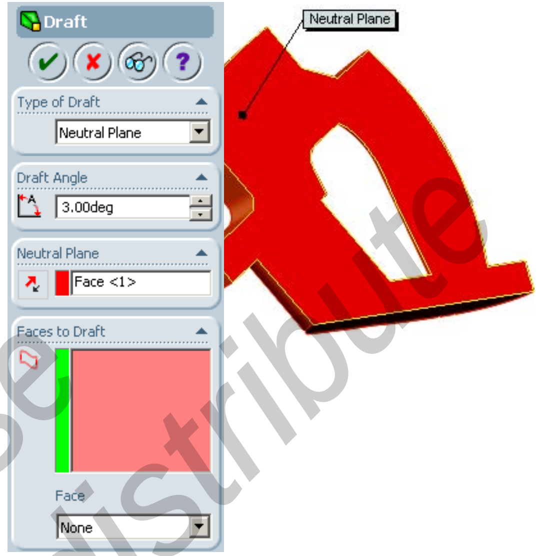
- From the **Insert** menu, choose **Features, Draft...**.
- Or, on the Features toolbar, click the **Draft** tool.

Draft Using a Neutral Plane

The process of adding draft requires the selection of one **Neutral plane** and one or more **Faces to draft**.

6 **Neutral plane draft.**
Click **Insert, Features, Draft...** and choose **Neutral Plane** as the **Type of Draft**.

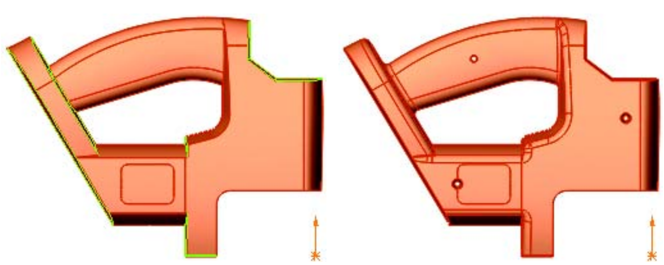
Select the rear planar face as the **Neutral Plane**. Set the **Draft Angle** to **3** degrees.



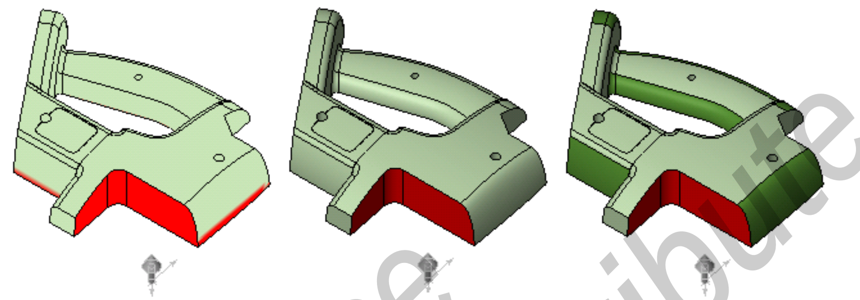
7 **Faces to draft.**
Select the eight faces to draft as shown at right.

Click **Reverse Direction** if necessary to point the arrow in the direction shown.

8 **Completed draft.**
Use **Roll to End** to rebuild all features of the model. The draft faces shown before (left) and after (right) rollback.



9 Draft analysis recheck.
Recheck **Draft Analysis** using the same **Direction of Pull** and **Draft Angle**.



Shelling

A shelling operation is used to “hollow out” a solid. You can apply different wall thicknesses to selected faces. You can select faces to be removed. In this example, all walls will have the same thickness: **2mm**.

Order of Operations

Most plastic parts have rounded corners. If you add fillets to the edges *before* shelling and the fillet radius is greater than the wall thickness, the inside corners of the part will automatically be rounded. The radius of the inside corners will equal the fillet radius minus the wall thickness. Taking advantage of this can eliminate the tedious task of filleting the inside corners.

If the wall thickness is greater than the fillet radius, the inside corners will be sharp.

Introducing: Insert Shell

Insert Shell removes selected faces and adds thickness to others to create a thin walled solid. It can create multiple thicknesses in the same shelling command.

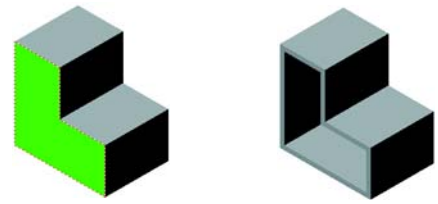
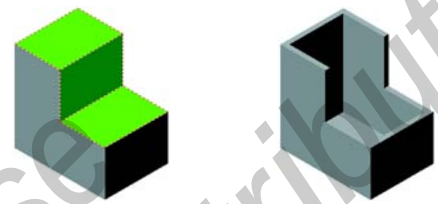
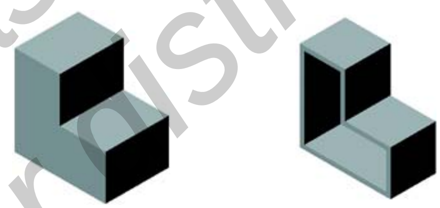
Where to Find It

- From the **Insert** menu choose **Features, Shell...**.
- Or, on the Features toolbar, click **Shell**.

Face Selection

Shelling can remove one or more faces from the model or create a fully enclosed void. Here are some examples:

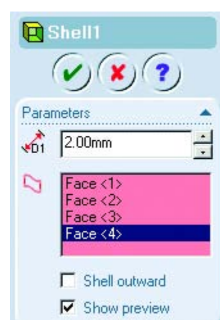
One face selected.	

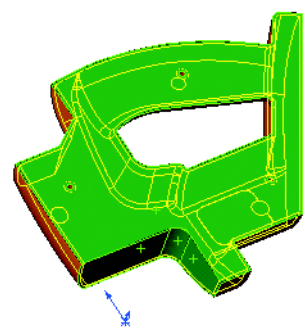
One face selected.	
Multiple faces selected.	
No faces selected. Note: The results are shown sectioned, using the **Section View** command.	

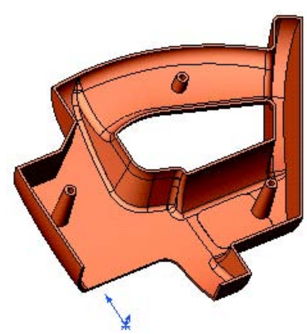
10 Shell.
Click **Insert, Features, Shell...** and set the **Thickness** to **2mm**.

Select the 4 faces shown as the **Faces to Remove** and check **Show preview**.

Click **OK**.







Tip Clear **Show preview** before selecting the faces, otherwise the preview will be updated with each selection, slowing down the operation.

Reference Planes

The **Plane Wizard** can be used to create a variety of reference planes using different geometry. Planes, faces, edges, vertices, surfaces and sketch geometry can all be used.

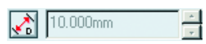
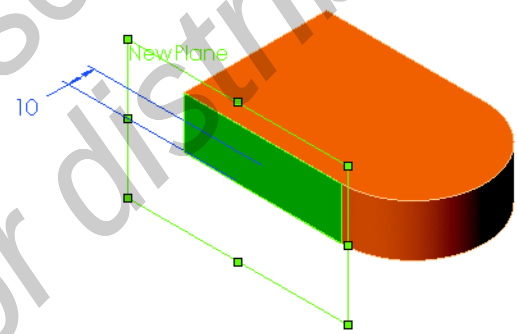
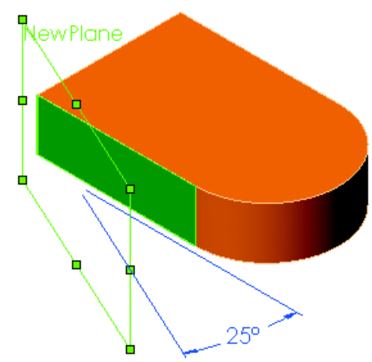
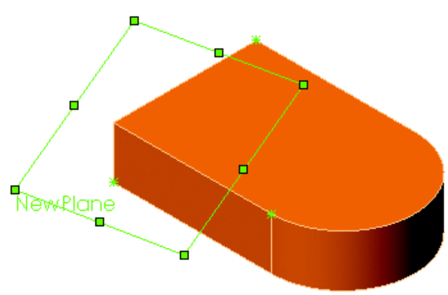
Where to Find It

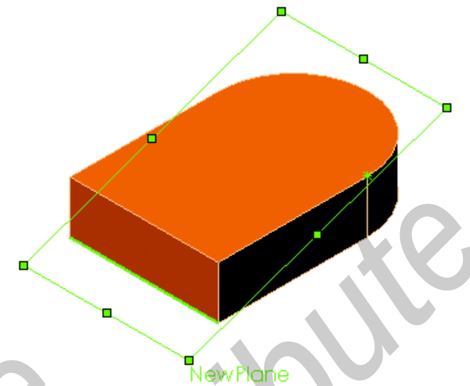
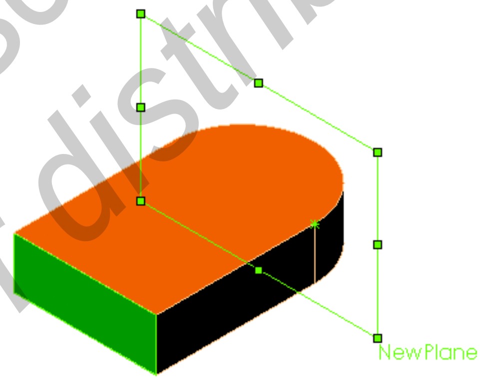
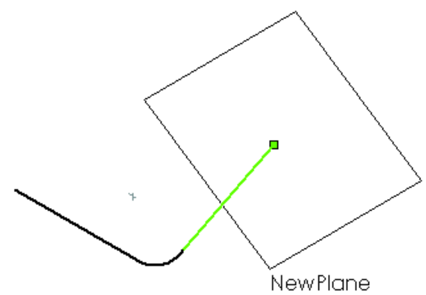
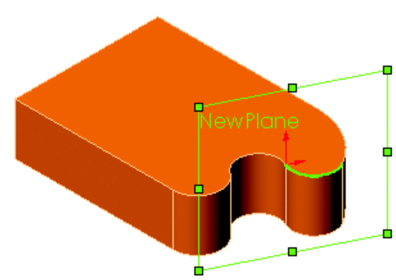
- Click **Plane** on the Reference Geometry toolbar.
- Or, click **Insert, Reference Geometry, Plane...**.

Shortcut

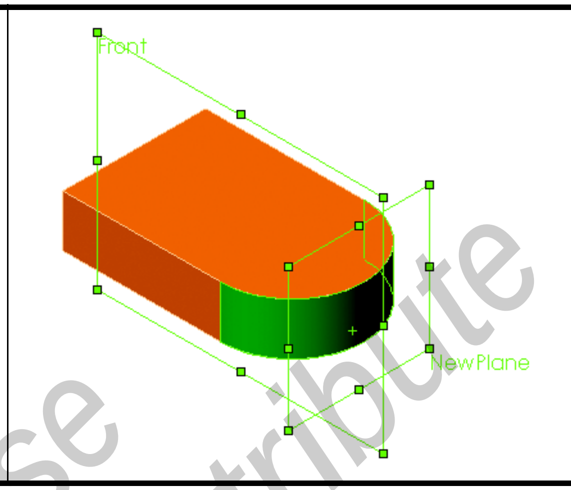
Press **Ctrl** and drag an existing reference plane to create an **Offset** plane.

Here are some examples:

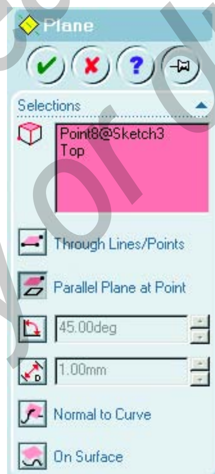
 Select a planar model face or plane.  Optionally create a series of parallel planes each this distance apart.	
 Select a planar model face (or plane) and edge or axis. Optionally create a series of angled planes this angle apart.	
 Select three vertices...	

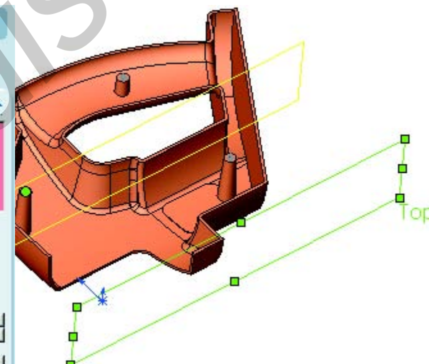
or a line and a vertex.	
Parallel Plane at Point Select a face and a vertex.	
Normal to Curve Select a sketched line and an endpoint.	
Shortcut to above: Select an edge and click **Insert, Sketch**.	

Select a cylindrical face and a reference plane.



11 Parallel to plane at a point.
Show the sketch of the `Stand Offs` feature. Click **Insert, Reference Geometry, Plane...** and click **Parallel Plane at Point**. Select the `Top` plane and the point indicated.





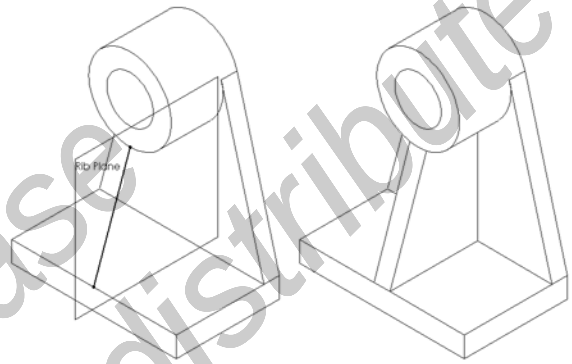
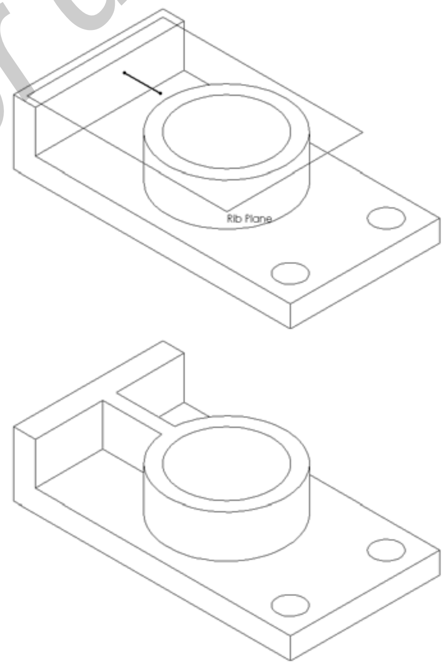
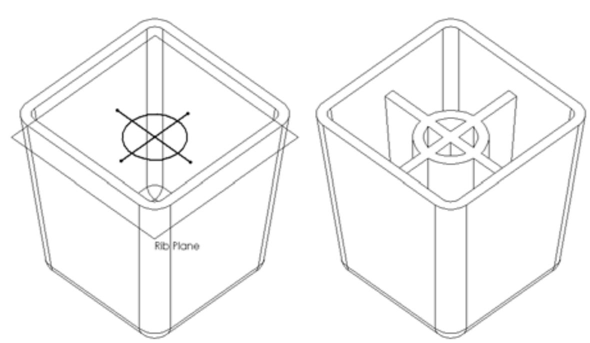
Rename the plane `thru standoff`.

Ribs

The rib tool, **Insert, Features, Rib...**, allows you to create ribs using minimal sketch geometry. The tool prompts you for the thickness, direction of the rib material, how you want to extend the sketch if necessary, and whether you want draft.

Rib Sketch

The rib sketch can be simple or complex. It can be as simple as a single sketched line that forms the rib centerline, or it can be more elaborate. Depending on the nature of the rib sketch, the rib can be extruded parallel or normal to the sketch plane. Simple sketches can be extruded either parallel to or normal to the sketch plane. Complex sketches can only be extruded normal to the sketch plane. Here are some examples:

Simple sketch extruded parallel to the sketch plane.	
Simple sketch extruded normal to the sketch plane.	
Complex sketch extruded normal to the sketch plane.	

Introducing: Insert Rib

Insert Rib creates a flat topped rib either with or without draft. The rib is based on a sketched contour line that defines the path of the rib. A full round fillet can be added to round off the rib.

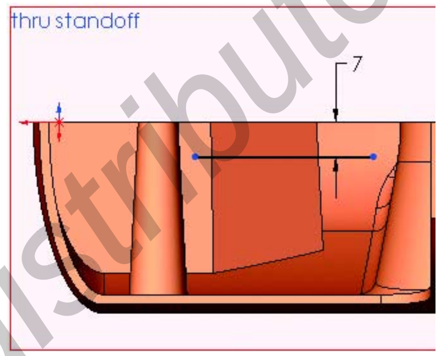
Where to Find It

- From the **Insert** menu choose **Features, Rib...**.
- Or the pick the **Rib** tool on the Features toolbar.

---

12 Sketch line.

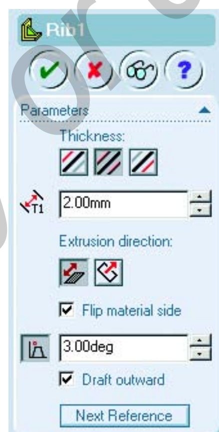
Create a new sketch on the `thru standoff` plane. Click the view **Normal To** and use the **Alt +Arrow** (left and right) to rotate the view.

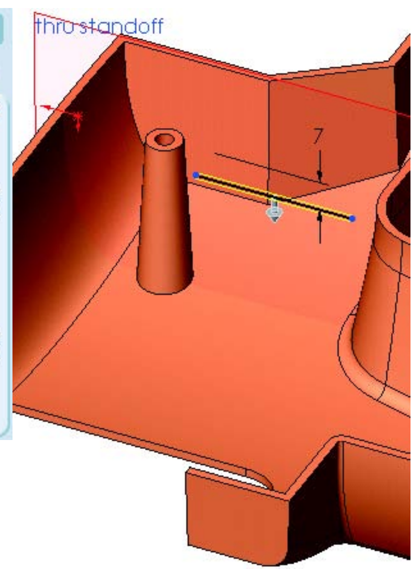


Sketch a line, underdefined and dimensioned as shown. Note that the line is **Vertical**.

13 Rib tool.

Click the **Rib** tool and set the parameters shown:





- **Thickness: 2mm** Create rib on **Both Sides** of sketch
- **Extrusion direction: Parallel to Sketch**
- **Draft : 3° outward**

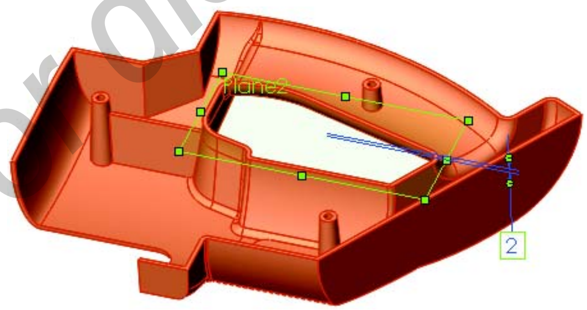
Look at the **Flip material side** arrow which indicates the direction the rib will be extruded. If necessary, reverse the direction.

14 Detailed preview.
Click **Detailed Preview** and clear **Highlight new or modified faces** to see a preview of the rib.

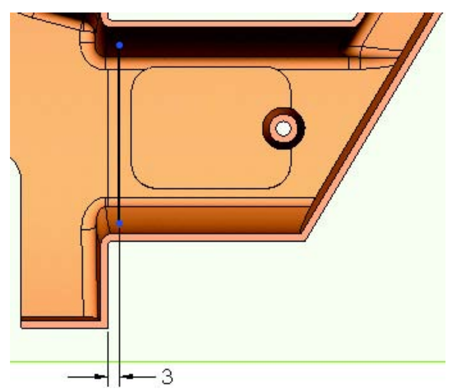
The completed rib extends down to the bottom face and along both ends of the sketched line.

Click **OK**.

15 Offset distance.
Select the thickness face and use the **Plane** tool to create a new reference plane **Offset** by **2mm** to the inside. Rename the plane `lin_patt`.



16 Sketch line.
Using the new plane, sketch a vertical line to represent a rib. Add the dimension but leave the sketch under defined.



Sketch Patterns

You can pattern sketch entities in either a linear or circular pattern. This is referred to as “step and repeat”. Once you create the pattern, the sketch entities are related with a **Patterned** relation. You can edit the definition of a step and repeat pattern once it is created.

Introducing: Linear and Circular Pattern

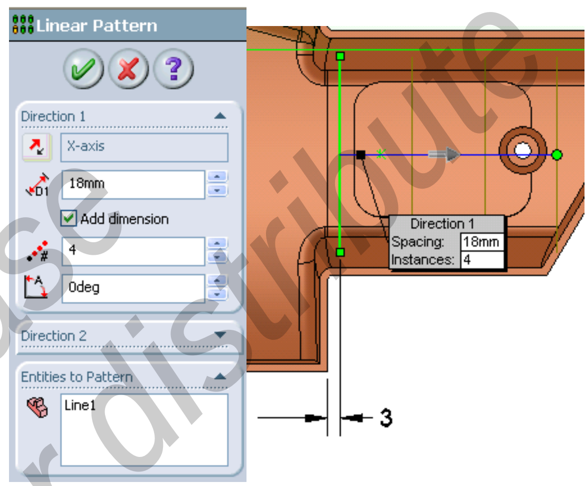
Sketch patterns are an efficient way to replicate sketch geometry without having to draw each entity. They are particularly useful for features such as ribs or as the basis of a feature pattern.

Where to Find It

- On the Sketch toolbar click **Linear Pattern** or **Circular Pattern** .
- Or click **Tools, Sketch Tools, Linear Pattern...** or **Circular Pattern...**.

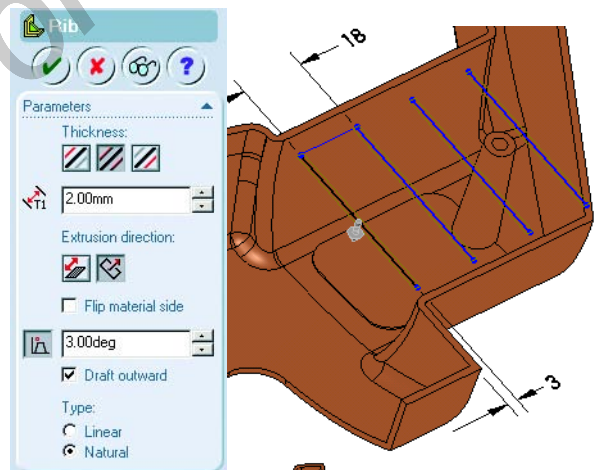
17 Copies.

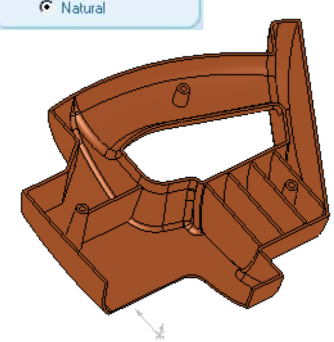
Click **Linear Pattern** and **Add dimension**. Add **4** copies of the line with the **Spacing 18mm**. The dimension is added automatically.



18 Rib.

Click the **Rib** tool. Click the **Normal to Sketch** option and point the arrow towards the material. For the remaining options, repeat those used in the previous rib feature.



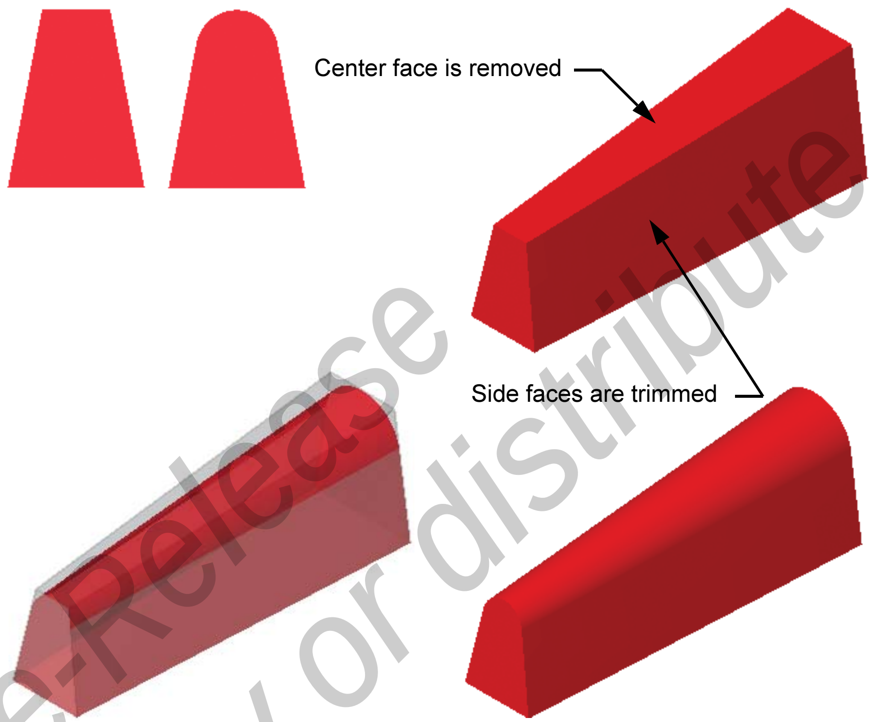


Full Round Fillets

The **Full Round Fillet** option creates a fillet that is tangent to three adjacent sets of faces. Each face set can contain more than one face. However, within each face set, the faces must be tangent continuously.

Introducing: Full Round Fillets

A full round fillet does not need a radius value. The radius is determined by the shape of the faces you select.

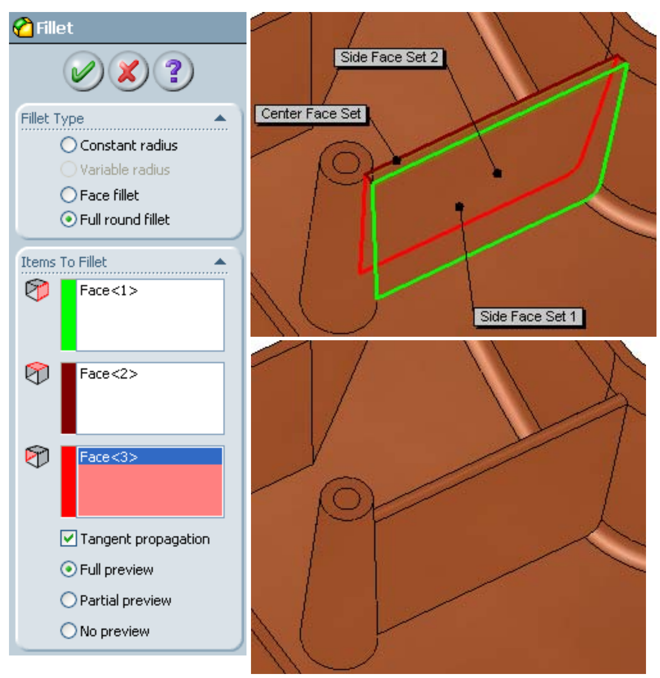


Where to Find It

- From the **Insert** menu choose **Features, Fillet/Round...**.
- Or click **Fillet** on the Features toolbar.

19 Full round.
Click Fillet icon and the **Full round fillet** option. Under **Items To Fillet**, select one face in each set as shown.

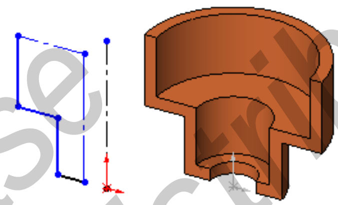
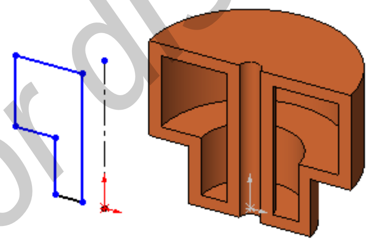
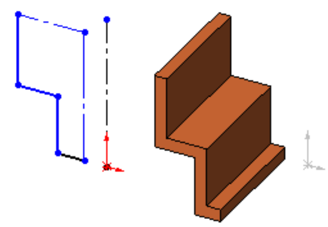
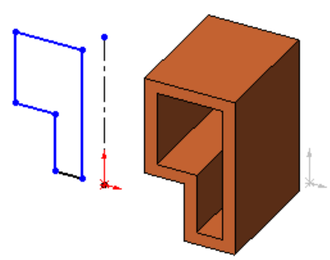
20 Save and close the file.



Thin Features

Thin Features are made by using an *open* sketch profile and applying a wall thickness. The thickness can be applied to the inside or outside of the sketch, equally on both sides of the sketch or unequally on either side. Thin feature creation is automatically invoked for open contours that are extruded or revolved. Closed contours can also be used to create thin features.

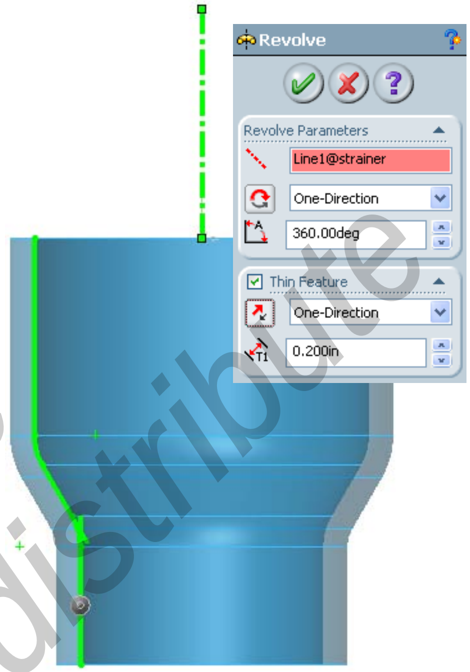
Thin features can be created for extrudes, revolves, sweeps and lofts.

Revolve, open	
Revolve, closed	
Extrude, open	
Extrude, closed	

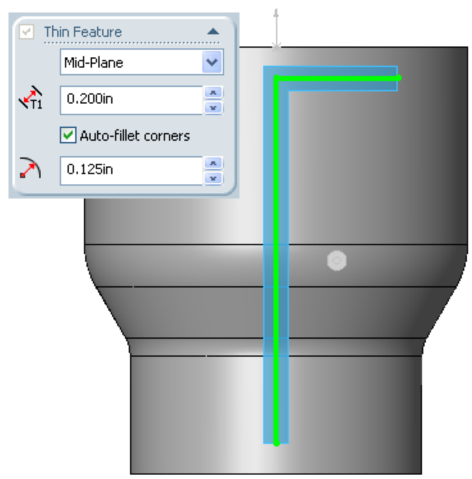
1 Open `Thin_Features`.

2 Thin revolve.
Select the `strainer` sketch and the **Revolve** tool. When the system asks whether the sketch should be automatically closed, click **No**.

Set the **Direction 1 thickness** to **0.20”** and the direction to the outside.

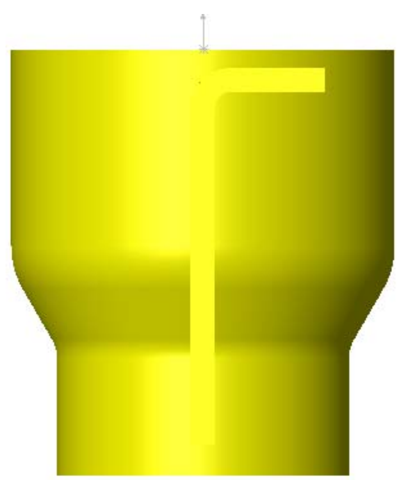


3 Thin extrude.
Select the `bracket` sketch and the **Extrude** tool. Set it to **Mid-Plane** and **0.20”**. Click Auto-fillet corner and set the **Fillet Radius** to **0.125”**.

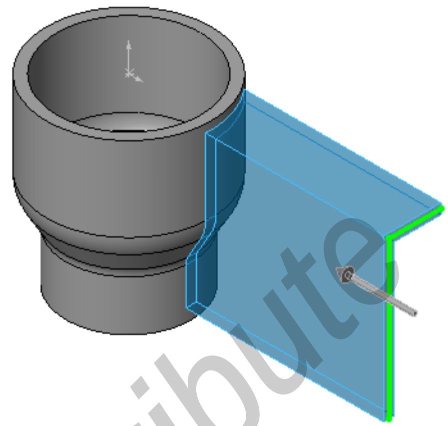


4 Preview.

Click **Detailed Preview** to view the auto fillets. Click the button again to dismiss the preview.

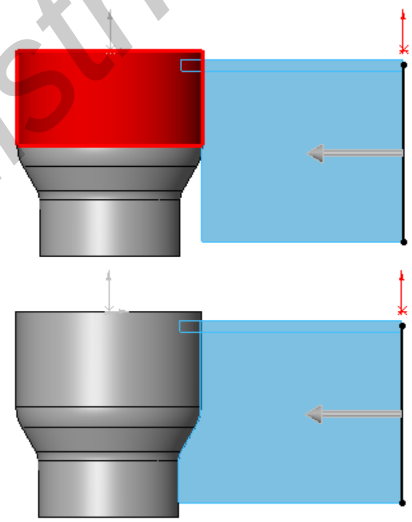


5 Direction.
Set the direction of the extrude towards the base feature and use **Up To Next**. Click **OK**.



Note

This example offers another comparison between **Up To Surface** (top) and **Up To Next** (bottom).



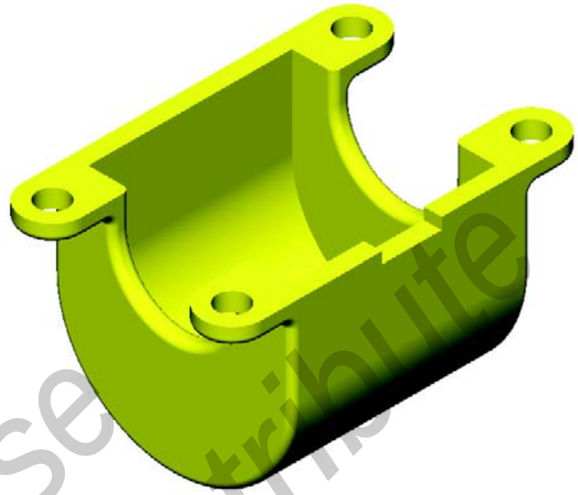
Exercise 46: Pump Cover

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- Sketching.
- Extrusions.
- Shelling.
- Mirroring features.

Units: **inches**



Design Intent

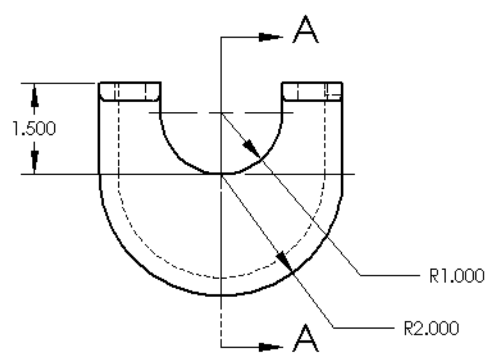
The design intent for this part is as follows:.

1. The tabs are all equal size and shape.
2. Holes in the tabs are all equal.
3. All fillets are equal at radius **0.12"**.
4. Wall thickness is constant.
5. Slot is centered on edge.
6. Excluding the slot, part is symmetrical about two planes.

Dimensioned Views

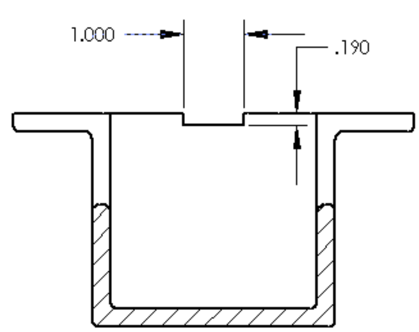
Use the following graphics with the design intent to create the part.

`Front view`



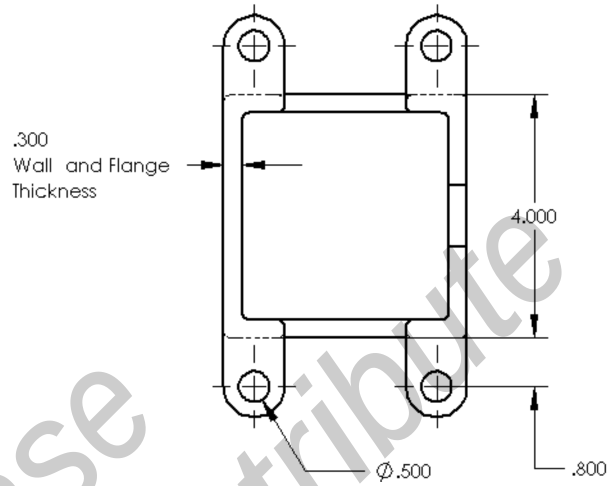
`Section A-A`

Slot detail.

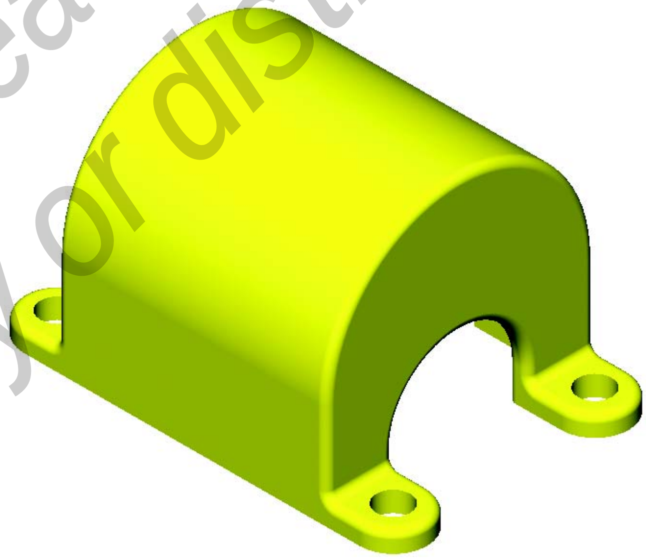


SECTION A-A

Top view



Reverse Isometric



Exercise 47: Ceiling Fan Ball

Create this part using the information and dimensions provided.

This lab reinforces the following skills:

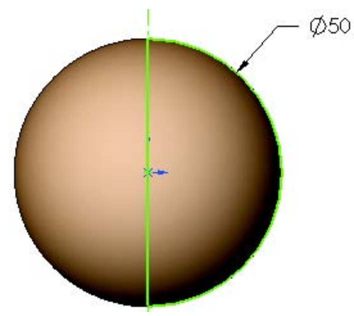
- Creating reference planes.
- **Cut with Surface**.
- **Offset Entities**.
- **Convert Entities**.
- Revolved bosses and cuts.

Procedure

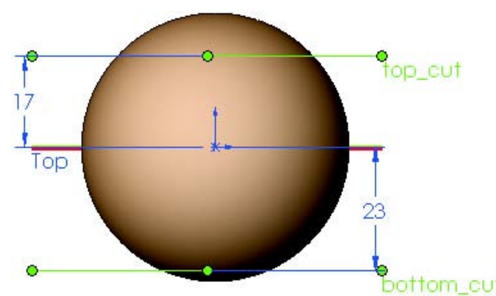
Use the `Part_MM` template.

1 Open a new part.

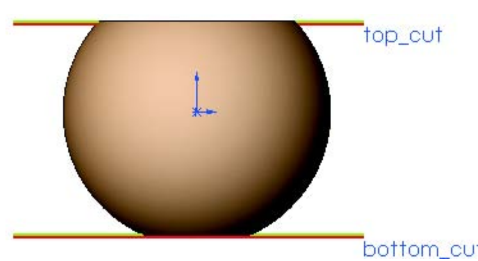
2 Sphere.
Create a sphere using a sketch and revolved feature.



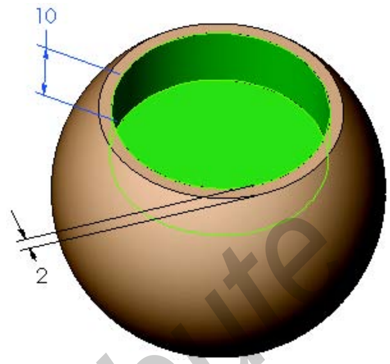
3 Offset planes.
Create new reference planes offset from `Top`.



4 Cut with surface.
Create one cut feature for each of the reference planes created.

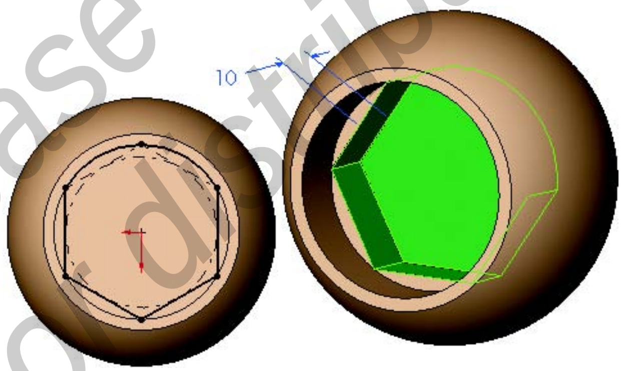


5 Offset entities.
Offset from the cut edge and create a cut using the geometry.

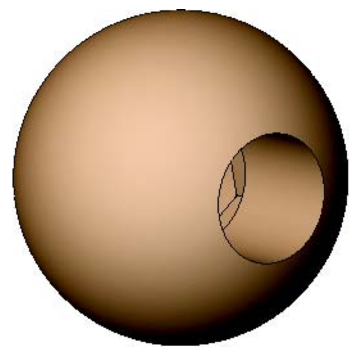


6 Polygon and convert entities.
Use the **Polygon** tool to create a hexagon and modify it using **Convert Entities**.

Create a **Cut**.

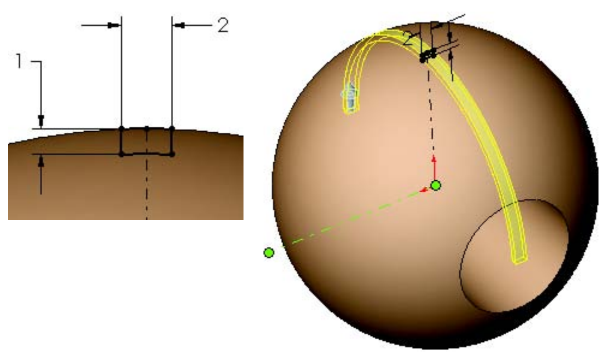


7 Converted edge.
Covert the edge created by the cut with surface to create a **Through All Cut**.



8 Revolved cut.
Use converted and offset edges to create a sketch profile.

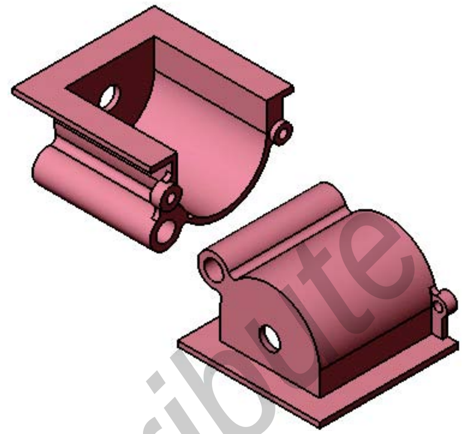
Revolve that sketch using **Mid-Plane** and and angle of **180°**.



9 Save and close the part.

Exercise 48: Motor Shield

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.



This lab uses the following skills:

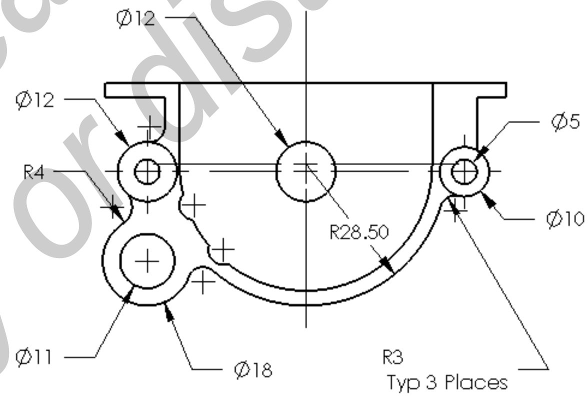
- Sketching.
- Extrusions.
- Shelling.

Units: **millimeters**

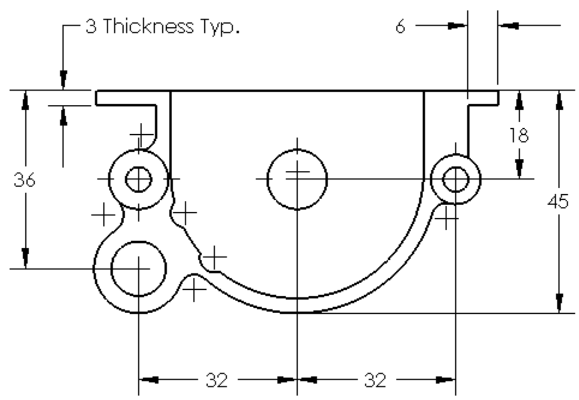
Dimensioned Views

Use the following graphics with the design intent to create the part.

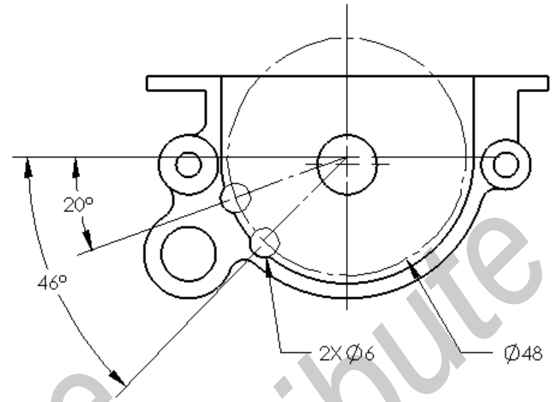
`Right` view: Radii and Diameters



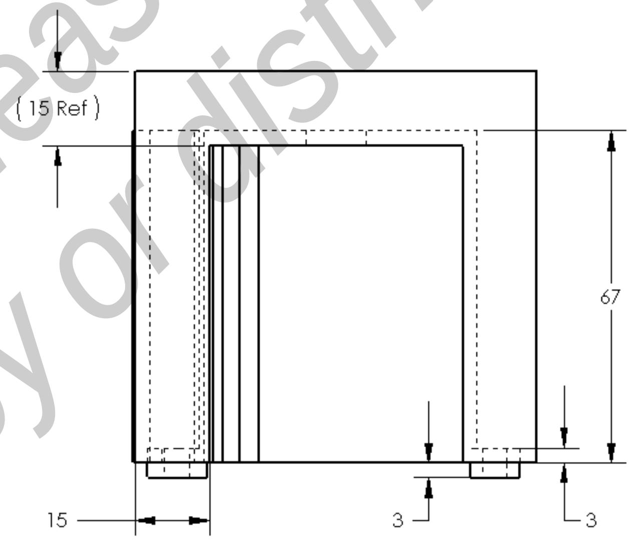
`Right` view: Locations



`Right` view: Interior cuts

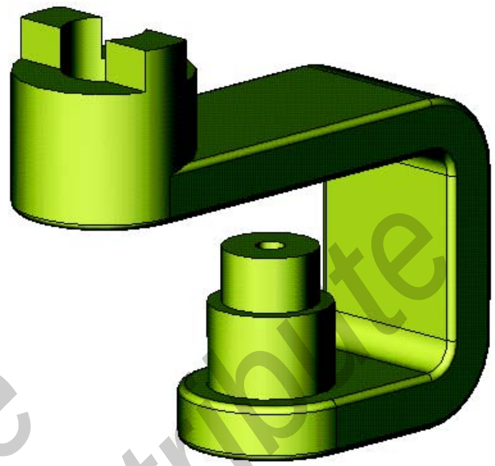


`Top` view



Exercise 49: Arm

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.



This lab uses the following skills:

- Sketching with symmetry.
- Thin features.
- Creating **Full round** fillets.
- Using the **Offset from surface** end condition.

Units: **inches**

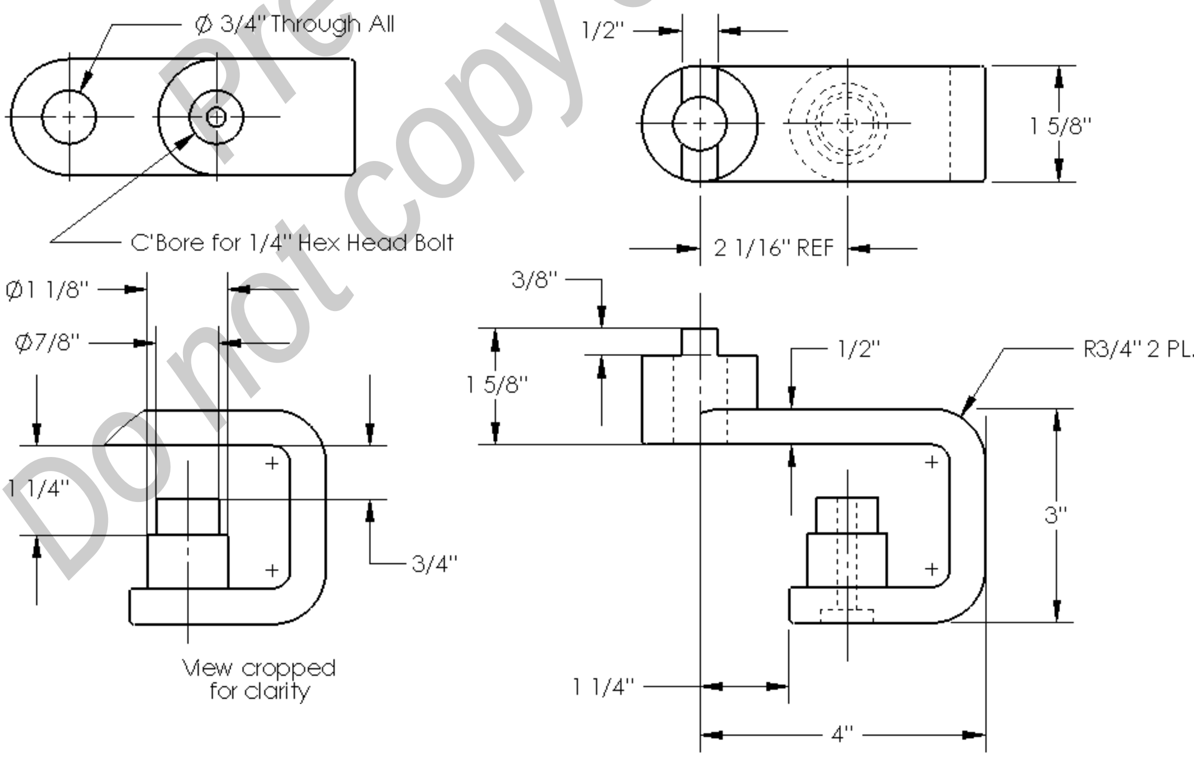
Design Intent

The design intent for this part is as follows:

1. Part is symmetrical.
2. All fillets and rounds **1/16"**.

Dimensioned Views

Use the following graphics with the design intent to create the part.



Exercise 50: Hook

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- **Up To Surface**.
- Thin features.
- **Full round** fillets.

Units: **mm**

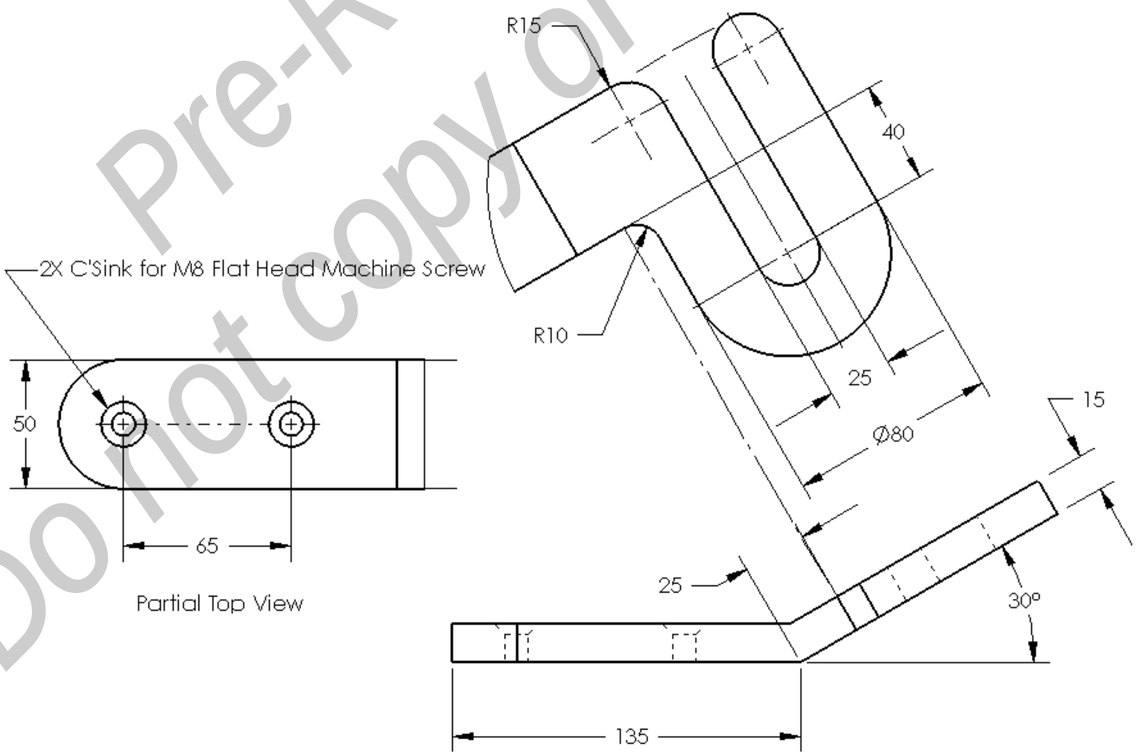
Design Intent

The design intent for this part is as follows:

1. Metal thickness is constant.
2. All chamfers **2mm X 2mm**.

Dimensioned Views

Use the following graphics with the design intent to create the part.



Exercise 51: Blow Dryer

Create this part by following the steps as shown.

This lab uses the following skills:

- Shelling.
- Ribs tool.
- Draft.
- Rollback.
- Linear patterns.
- Full round fillets.
- Hole wizard.

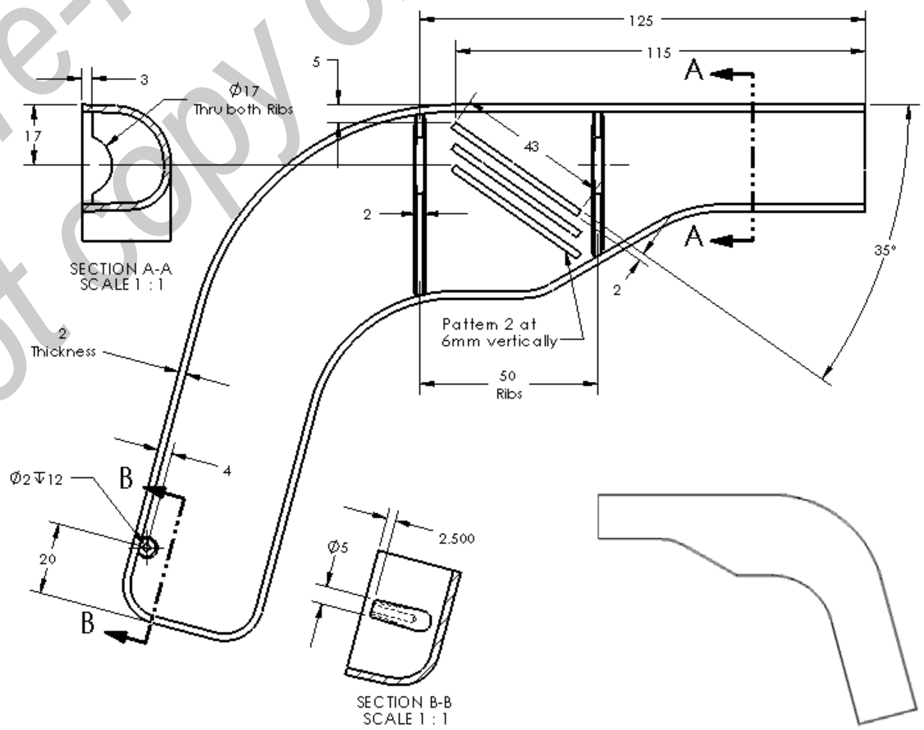
Procedure

Open an existing part in the `Exercises` folder.

1 **Open the part `Blow Dryer`.**

2 **Complete the part.**
Complete the part using the following guidelines.

- Wall thickness is constant.
- Vents and ribs are the same size.
- All fillets and rounds **1mm** except full rounds on ribs.
- Draft is **2** degrees. No draft on outlet face.



Note The draft feature should precede the existing fillets in the part.

3 **Save and close the part.**

Exercise 52: Face Shield

Create this part using the information and dimensions provided.

This lab reinforces the following skills:

- Rollback.
- Creating reference planes.
- Creating reference axes.
- Cut with Surface.
- Hole Wizard holes.

Procedure

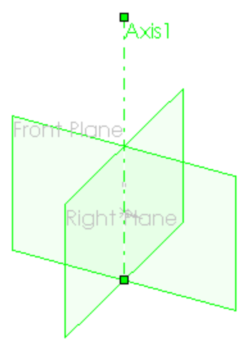
Open an existing part in the `Exercises` folder.

1 **Open the part `Face Shield`.**

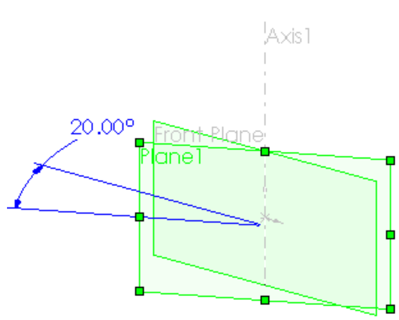
2 **Rollback.**
Drag the rollback bar up to a position before the `Initial Glass` feature.

The new reference geometry can be placed before or after the feature. They will placed at the earliest possible position, before the feature.

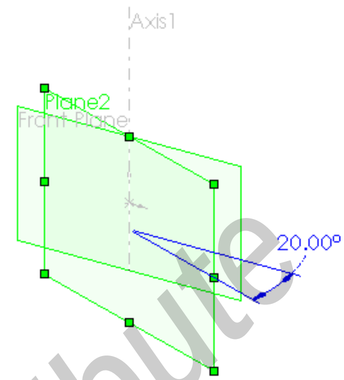
3 **`Axis1`.**
Create `Axis1` as the intersection of the `Front` and `Right` planes.



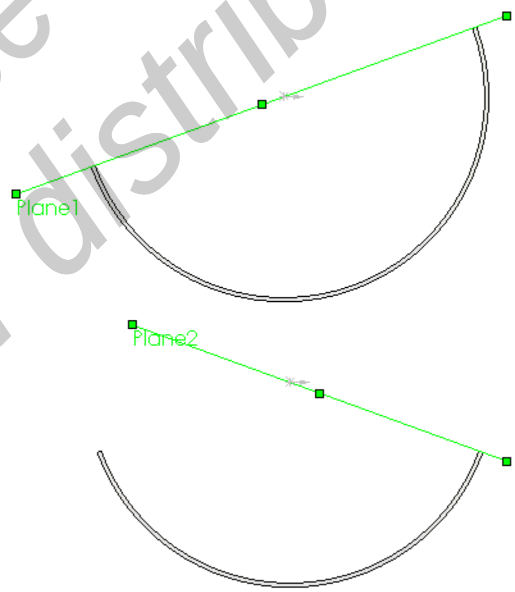
4 **`Plane1`.**
Create new reference plane **At Angle** of **20°** from `Front` and about `Axis1`.



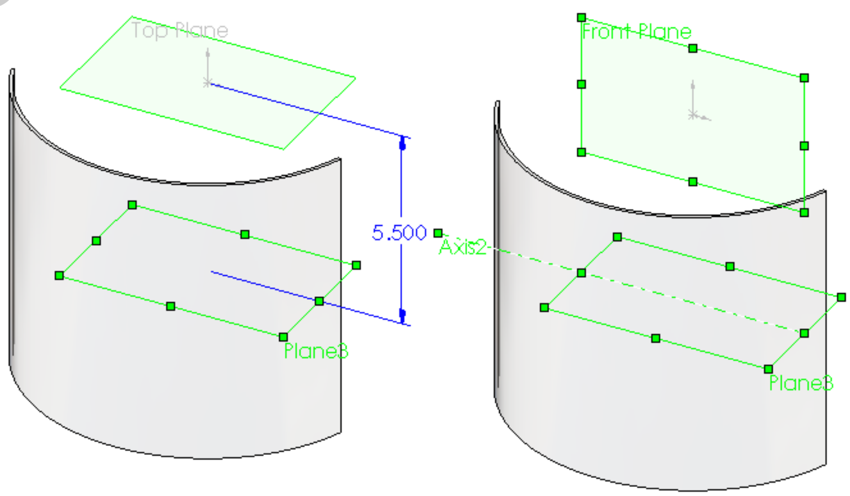
5 Plane2.
Create Plane2 using a similar procedure as the previous plane.



6 Cuts with planes.
Cut the model geometry in sequence using reference planes Plane1 and Plane2.

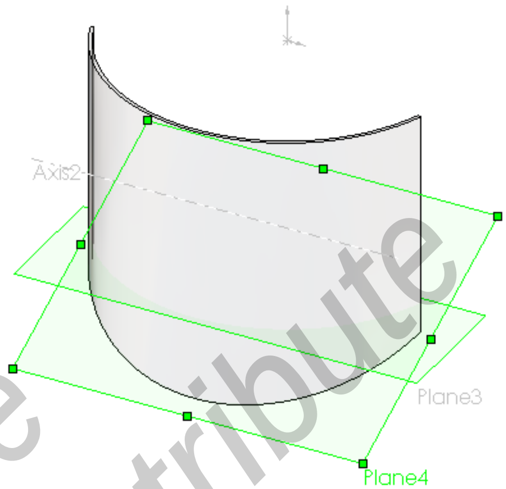


7 Offset plane and axis.
Create the Plane3 as an offset from Top. Use Front and Plane3 to create Axis2.



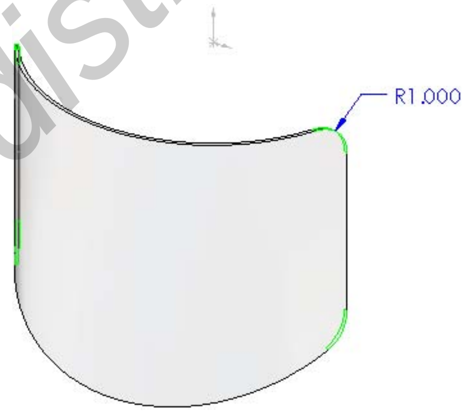
8 Plane4.

Create `Plane4` as an angle of **25°** from `Plane3` about `Axis2`. Cut the model using that plane.



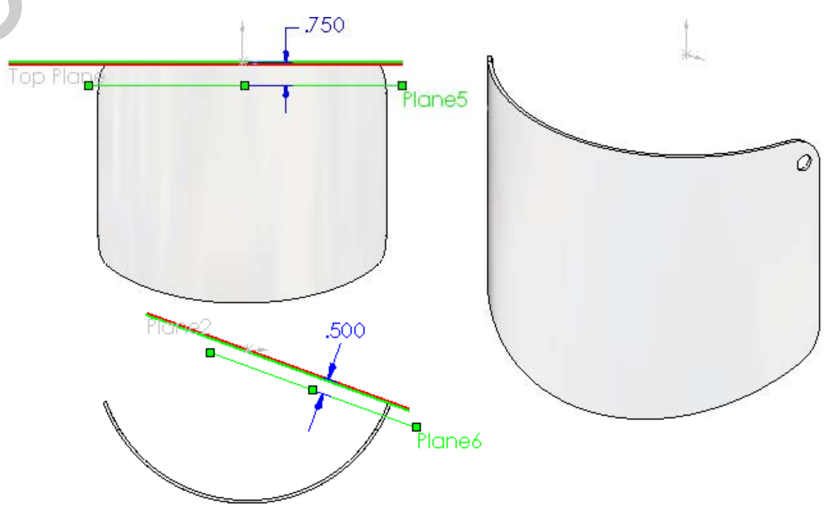
9 Rounds.

Add rounds of radius **1"** to all four corners.



10 Hole wizard.

Create the reference planes `Plane5` and `Plane6` as shown. Add the `7/16 (0.4375) Diameter Hole` to the outer face and locate the center using the new planes. Mirror the hole feature.



11 Save and close the part.

Lesson 12
Bottom-Up Assembly Modeling

Upon successful completion of this lesson, you will be able to:

- Create a new assembly.
- Insert components into an assembly using all available techniques.
- Add mating relationships between components.
- Utilize the assembly-specific aspects of the FeatureManager design tree to manipulate and manage the assembly.
- Insert sub-assemblies.
- Use part configurations in an assembly.

Case Study: Universal Joint

This lesson will examine assembly modeling through the construction of a universal joint. The joint consists of several components and one sub-assembly.

Bottom-Up Assembly

Bottom-Up assemblies are created by adding and orienting existing parts in an assembly. Parts added to the assembly appear as *Component Parts*. Component parts are oriented and positioned in the assembly using **Mates**. Mates relate faces and edges of component parts to planes and other faces/edges.

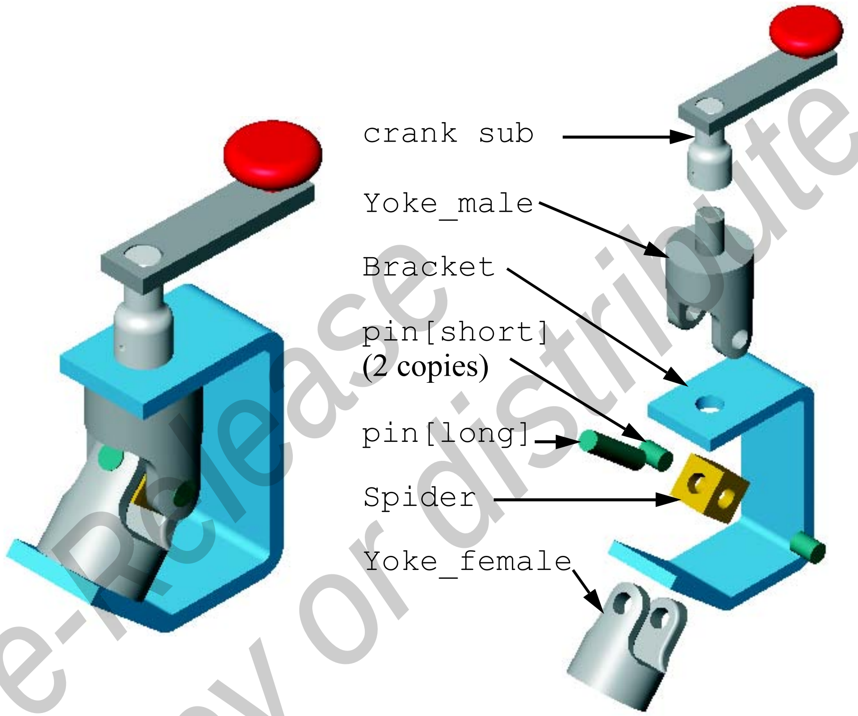
Stages in the Process

Some key stages in the modeling process of this part are shown in the following list. Each of these topics comprises a section in the lesson.

- **Creating a new assembly**
  New assemblies are created using the same method as new parts.
- **Adding the first component**
  Components can be added in several ways. They can be dragged and dropped from an open part window or opened from a standard browser.
- **Position of the first component**
  The initial component added to the assembly is automatically fixed as it is added. Others components can be positioned after they are added.
- **FeatureManager design tree and symbols**
  The FeatureManager includes many symbols, prefixes and suffixes that provide information about the assembly and the components in it.
- **Mating components to each other**
  Mates are used to position and orient components with reference to each other. Mates remove degrees of freedom from the components.
- **Sub-assemblies**
  Assemblies can be created and inserted into the current assembly. They are considered sub-assembly components.

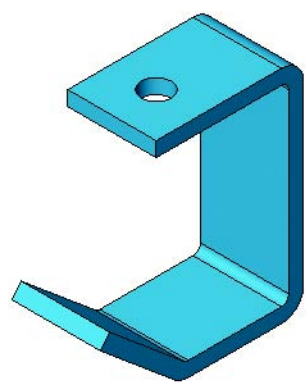
The Assembly

In this lesson we will make an assembly using existing components. The assembly is a universal joint, and is made up of a number of individual parts and one sub-assembly as shown below:



1 Open an existing part.
Open the part `bracket`. A new assembly will be created using this part.

The first component added to an assembly should be a part that will not move. By fixing the first component, others can be mated to it without any danger of it moving.



Creating a New Assembly

New assemblies can be created directly or be made from an open part or assembly. The new assembly contains an origin, the three standard reference planes and a special feature.

Introducing: Make Assembly from Part/ Assembly

Use the **Make Assembly from Part/Assembly** option to generate a new assembly from an open part. The part is used as the first component in the new assembly and is fixed in space

Where to Find It

- Click **Make Assembly from Part/Assembly** on the Standard toolbar.
- Or, click **File, Make Assembly from Part**.

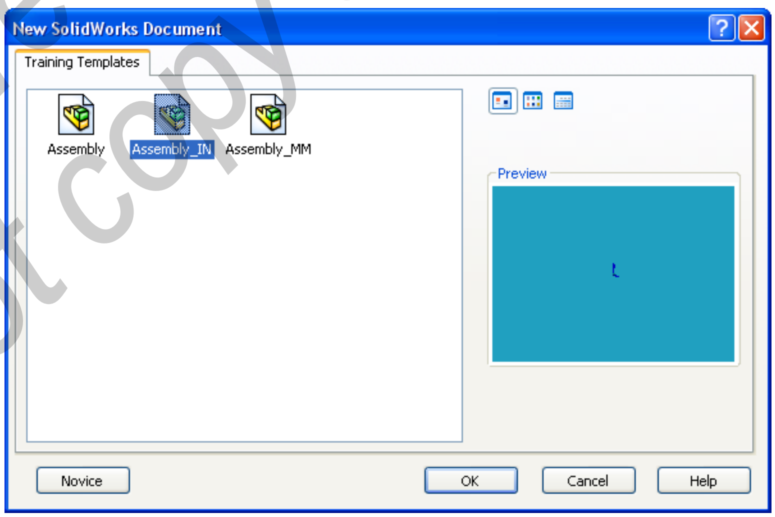
Introducing: New Assembly

Create a new assembly file using a template.

Where to Find It

- Click **New** on the Standard toolbar.
- Or, click **File, New...**.

2 **Choose template.**
Click **File, Make Assembly from Part** and select the **Advanced** button from the **New SolidWorks Document** dialog. Select the Training Template `Assembly_IN`.



Shortcut

Double-click the desired template to automatically open a new assembly document using that template.

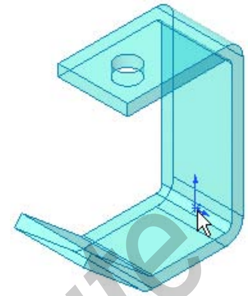
Note

The units of the assembly can be different from the units of the parts. For example, you can assemble a mixture of inch and millimeter parts in an assembly whose units are feet. However, when you edit the dimensions of *any* of the parts in the context of the assembly, they will be displayed in the units of the assembly, not those of the part itself. Using **Tools, Options...**, you can check the units of the assembly and if desired, change them.

3 Locate component.
Place the component at the origin using the cursor over the origin symbol. The part will appear in the assembly FeatureManager design tree as **Fixed (f)**.

4 Save.
Save the assembly under the name `Universal Joint`. Assembly files have the file extension `*.sldasm`.

Close the `bracket` part file.



Position of the First Component

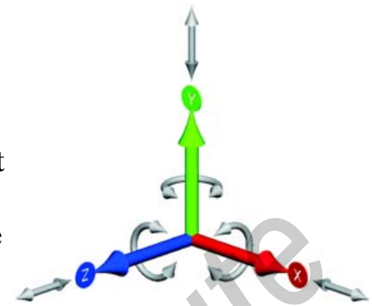
The initial component added to the assembly is, by default, **Fixed**. Fixed components cannot be moved and are locked into place wherever they fall when you insert them into the assembly. By using the cursor during placement, the component's origin is at the assembly origin position. This also means that the reference planes of the component match the planes of the assembly, and the component is fully defined.

Consider assembling a washing machine. The first component logically would be the frame onto which everything else is mounted. By aligning this component with the assembly's reference planes, we would establish what could be called "product space". Automotive manufacturers refer to this as "vehicle space". This space creates a logical framework for positioning all the other components in their proper positions.

FeatureManager Design Tree and Symbols

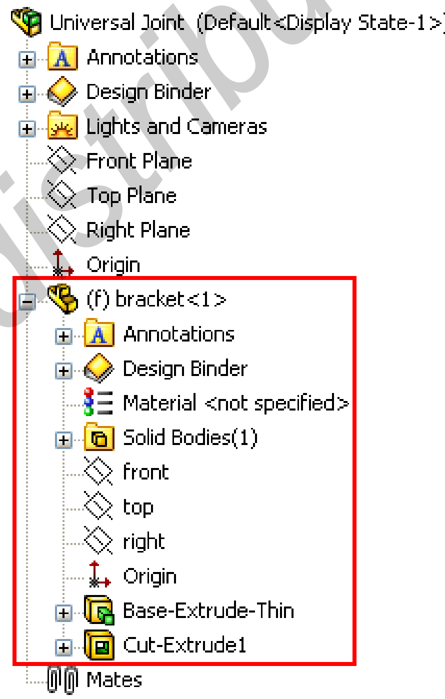
Within the FeatureManager design tree of an assembly, the folders and symbols are slightly different than in a part. There are also some terms that are unique to the assembly. Now that some parts and mates are listed there, they will be described.

Degrees of Freedom



There are six degrees of freedom for any component that is added to the assembly before it is mated or fixed: translation along the X, Y, and Z axes and rotation around those same axes. How a component is able to move in the assembly is determined by its degrees of freedom. The **Fix** and **Insert Mate** options are used to remove degrees of freedom.

Components



Parts that are inserted into the assembly, such as the `bracket`, are represented by the same top-level icon as is used in the part environment. Assemblies can also be inserted and are shown with a single icon. However, when the listing of these icons is expanded, the individual components and even the component's features are listed and accessible.

- **State of the component.**
  The part can be fully, over or under defined. A (+) or (-) sign in parentheses will precede the name if it is **Over** or **Under Defined**. Parts that are under defined have some degrees of freedom available. Fully defined ones have none.
  The **Fixed** state (**f**) indicates a component is fixed in its current position, but not mated. The question mark (**?**) symbol is for components that are **Not Solved**. These components cannot be placed using the information given.

- **Instance Number.**
  The instance number indicates how many copies of a certain component part are found in the assembly. The name `bracket<1>` indicates that this is the first instance of the `bracket`.

- **Component Part Folder.**
  Each component part contains the entire contents of the part, including all features, planes and axes.

Annotations

The `Annotations` feature is used for the same purpose as in a part. Annotations can be added at the assembly level and imported to a drawing. Their display is also controlled by the **Details** option.

Rollback Marker

The **Rollback** marker can be used in an assembly to rollback:

- **Assembly planes, axes, sketches**
- **Mates folder**
- **Assembly patterns**
- **In-context part features**
- **Assembly features**

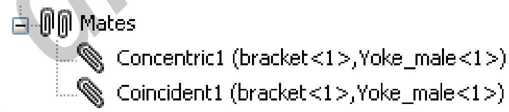
Any features below the marker are suppressed. Individual components cannot be rolled back.

Reorder

Certain objects in an assembly can be reordered. They are:

- **Components**
- **Assembly planes, axes, sketches**
- **Assembly patterns**
- **In-context part features**
- **Mates within the `Mates` folder**
- **Assembly features**

Mate Groups



The mating relationships in assemblies are grouped together into a **Mate Folder** named `Mates`. A mate group is a collection of mates that get solved in the order in which they are listed. All assemblies will have a mate group.

- **`Mates` Folder**
  The folder used to hold mates that are solved together. Identified by a double paper clip icon.
- **Mate**
  The relationships between faces, edges, planes, axes or sketch geometry that define the location and orientation of components. They are 3D versions of the 2D geometric relations in a sketch. Mates can be used to fully define a component that does not move, or under define one that is intended to move. Under no conditions should a component be over defined. The possible states for a mate are **Under Defined**, **Over Defined**, **Fully Defined** or **Not Solved**.

Adding Components

Once the first component has been inserted and fully defined, other parts can be added and mated to it. In this example, the `Yoke_male` part will be inserted and mated. This part should be under defined so that it is free to rotate.

There are several ways to add components to the assembly:

- **Use the Insert dialog.**
- **Drag them from the Explorer.**
- **Drag them from an open document.**

All these methods will be demonstrated in this lesson, beginning with use of **Insert Component**. This is the same dialog that appears automatically when **Make Assembly from Part** is used.

Insert Component

The **Insert Component** dialog is used to find, preview and add components to the current assembly. Click the **Keep Visible** (pushpin) button to add multiple components or multiple instances of the same component.

Where to Find It

- Click **Existing Part/Assembly** on the Assembly toolbar.
- Or, click **Insert, Component, Existing Part/Assembly...**.

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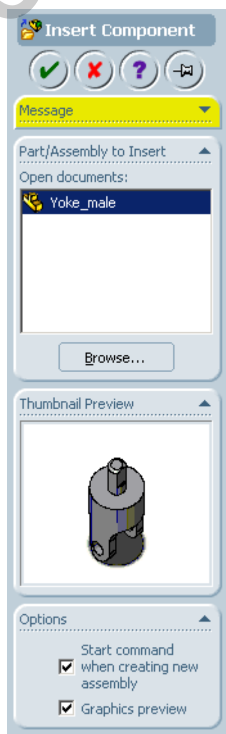
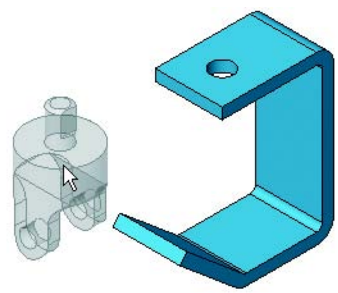
5 Insert `Yoke_male`.
Click **Insert, Component, Existing Part/Assembly...** and select the `Yoke_male` using the **Browse...** button. Position the component on the screen and click to place it.

The new component is listed as:

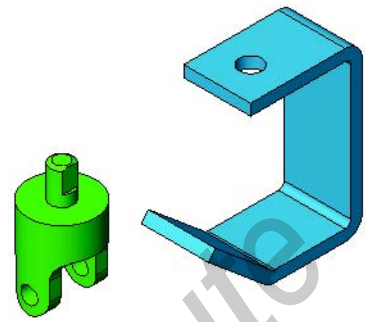
`(-) Yoke_male <1>`

This means that the component is the first instance of `Yoke_male` and it is under defined. It still has all six degrees of freedom.





6 Highlighting.
Clicking on a component in the FeatureManager design tree will cause that component to highlight (light green). Also, moving the cursor to a component in the graphics window will display the feature name.



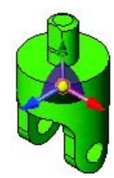
Moving and Rotating Components

One or more selected components can be moved or rotated to reposition them for mating using the mouse or the **Move** and **Rotate Component** commands. Also, moving under defined components simulates movement of a mechanism through dynamic assembly motion.

Where to Find It

Using the mouse:

- Drag and drop a component.
- Right-click a component, and select **Move with Triad**. Use the triad to move or rotate components along or around axes. Float over arrowhead: left-drag to move along the axis, right-drag to rotate about the axis.



Using the menus:

- From the pull-down menu choose: **Tools, Component, Rotate** or **Move**.
- Right-click the component, and select **Move...**.
- Or, on the Assembly toolbar pick one of these tools:

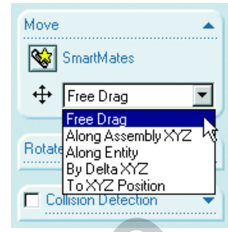
 Moves a component. This can also be used to rotate components that have rotational degrees of freedom.

 Rotates the component in one of several ways: about its centerpoint; about an entity such as an edge or axis; or by some angular value about the assembly X, Y, or Z axes.

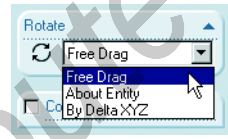
Note

Move Component and **Rotate Component** behave as a single, unified command. By expanding either the **Rotate** or **Move** options, you can switch between the two commands without ever closing the PropertyManager.

The **Move** tool has several options for defining the type of movement. The option **Along Entity** has a selection box, **Along Assembly XYZ**, **By Delta XYZ**, and **To XYZ Position** require coordinate values.

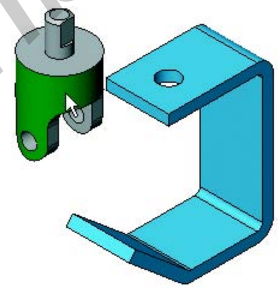


The **Rotate** tool also has options to define how the component will rotate.



7 **Move.**
Click on the component and drag it to move it closer to where it will be mated.

Other options for moving and rotating the component will be discussed later in this lesson.



Mate to Another Component

Obviously dragging a component is not sufficiently precise for building an assembly. Use faces and edges to mate components to each other. The parts inside the `bracket` are intended to move, so make sure that the proper degree of freedom is left available.

Note

The **Standard Mates** are discussed in this lesson. The **Advanced Mates** (Symmetric, Cam, Gear and Distance/Angle Limit Mates) are discussed in the *Advanced Assembly Modeling* manual.

Introducing: Insert Mate

Insert Mate creates relationships between component parts or between a part and the assembly. Two of the most commonly used mates are **Coincident** and **Concentric**.

Mates can be created using many different objects. You can use:

- Faces
- Planes
- Edges
- Vertices
- Sketch lines and points
- Axes and origins

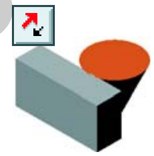
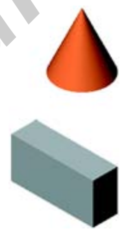
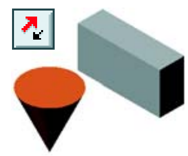
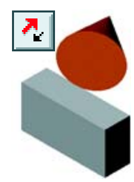
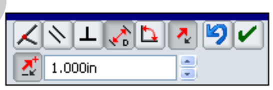
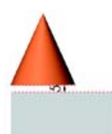
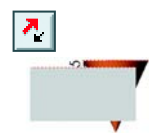
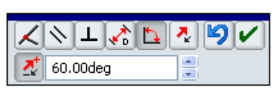
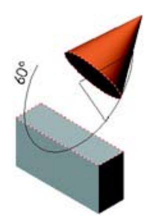
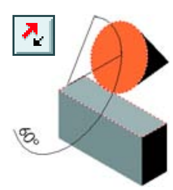
Mates are made between a *pair* of objects.

Where to Find It

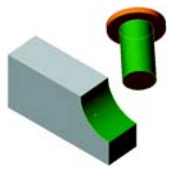
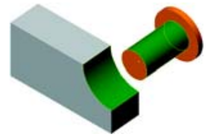
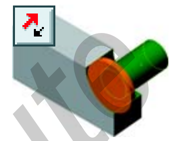
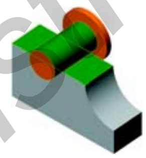
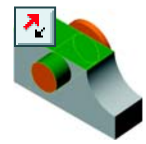
- On the **Insert** menu, select **Mate...**.
- Or, on the Assembly toolbar, click **Mate** .
- Or, right-click a component and choose **Add/Edit Mates**.

Mate Types and Alignment

Mates are used to create relationships between components. Faces are the most commonly used geometry in mates. The type of mate, in combination with the conditions **Anti-aligned** or **Aligned**, determines the result.

	**Anti-Aligned**	**Aligned**
Coincident (faces lie on the same imaginary infinite plane) 		
Parallel 		
Perpendicular Aligned and anti-aligned do not apply to **Perpendicular**. 		
Distance 		
Angle 		

Fewer options are available with cylindrical faces but they are every bit as important.

	Anti-Aligned	Aligned
Concentric  		
Tangent  		

Common Buttons

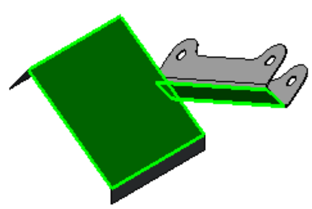
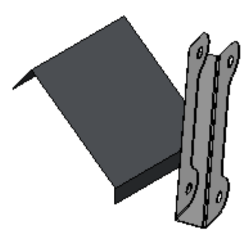
There are three buttons common to all the controls:

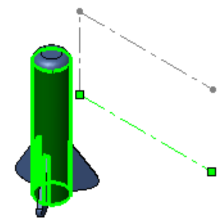
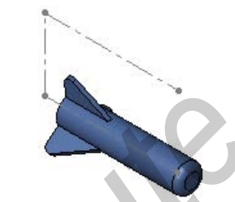
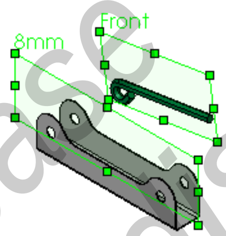
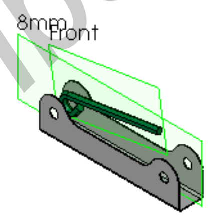
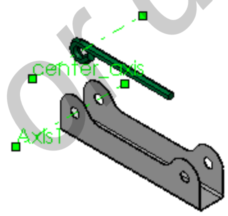
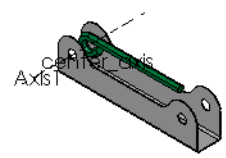
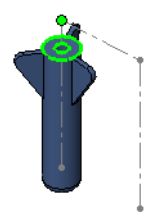
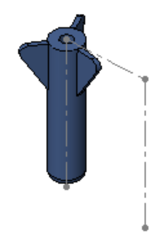
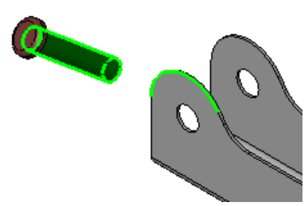
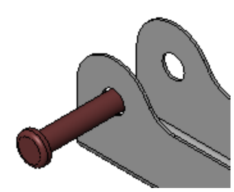
- is **Undo**
- is **Flip Mate Alignment**
- is **OK** or **Add/Finish Mate**.

In addition to these, the **Mate** dialog itself also has specific mate alignment controls, and .

Things you can mate to

There are many types of topology and geometry that can be used in mating.The selections can create many mates types.

Topology/ Geometry	Selections	Mate
Faces or Surface		

Topology/ Geometry	Selections	Mate
Line or Linear Edge		
Plane		
Axis or Temporary Axis		
Point, Vertex or Origin		
Arc or Circular Edge		

Tip

Although planes can be selected on the screen if they are visible, it is often easier to select them by name through the FeatureManager. Click the **"+"** symbol to see the tree and expand individual components and features.

Mating Concentric and Coincident

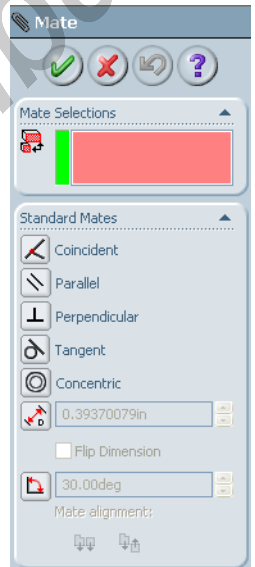
The `Yoke_male` component is to be mated so that its shaft aligns with the hole and the flat face contacts the `bracket` inner face. **Concentric** and **Coincident** mates will be used.

8 Selection filter.

The selection filter option is very useful in mating. Since many mates require face selections, set the **Select** option to faces. Note that this filter will remain in effect until SolidWorks or the part is exited, or the filter is changed.

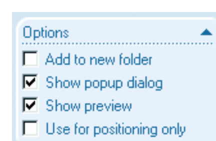
9 Mate PropertyManager.

Click on the **Insert Mate** tool to access the PropertyManager. If the PropertyManager is open, you can select the faces without using the **Ctrl** key.



Mate Options

Several mate options are available for all mates:



- **Add to new folder**
  Creates a new folder to hold all the mates created while the **Mate** tool is active. The folder resides in the `Mates` folder and can be renamed.
- **Show pop-up toolbar**
  Toggles the Mate Pop-up Toolbar on and off.
- **Show preview**
  Shows the positioning created by the mate as soon as the second selection is made. It is not finalized until the dialog **OK** is clicked.
- **Use for positioning only**

  This option can be used to position geometry without constraining it. No mate is added.

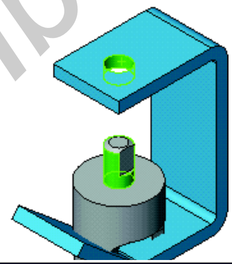
Introducing: Mate Pop-up Toolbar

The **Mate Pop-up** Toolbar is used to make selections easier by displaying the available mate types on the screen. The mate types that are available vary by geometry selection and mirror those that appear in the PropertyManager. The dialog appears on the graphics but can be dragged anywhere.

Either the on-screen or PropertyManager dialog can be used. This lesson uses the on-screen dialog. All types are listed in the chart *Mate Types and Alignment* on page 386.

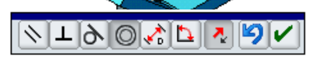
10 Selections and preview.

Select the faces of the `Yoke_male` and the `bracket` as indicated.



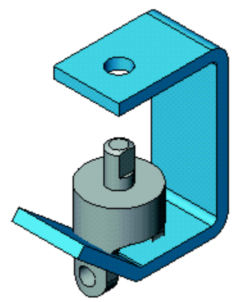
As the second face is selected, the **Mate Pop-up** Toolbar is displayed.

Concentric is selected as the default and the mate is previewed.



11 Add a mate.

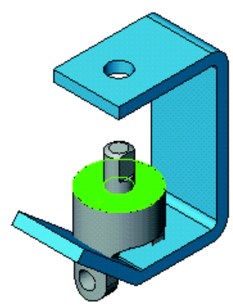
The faces are listed in the **Mate Settings** list. Exactly two items should appear in the list.



Accept the **Concentric** mate and click **Add/Finish Mate** (check mark).

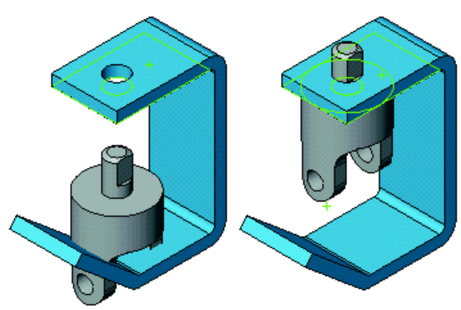
12 Planar face.

Select the top planar face of the `Yoke_male` component.

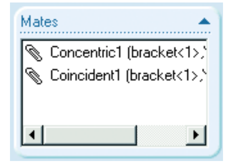


13 Select Other.

Use **Select Other** to select the hidden face of the `bracket` on the underside of the top flange. Add a **Coincident** mate to bring the selected faces into contact.



14 Mates listed.

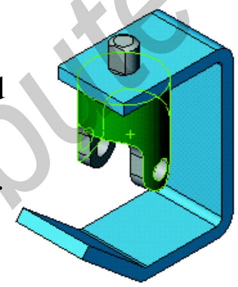


The mates, concentric and coincident, remain listed in the **Mates** group box. They will be added to the `Mates` folder when the **OK** button on the PropertyManager dialog is clicked. They can also be removed from this group box so that they are *not* added. Click **OK**.

15 State of constraint.

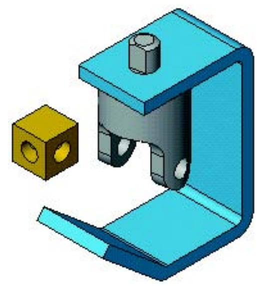
The `Yoke_male` component is listed as under constrained. It is still able to move by rotating around the axis of its cylindrical surface.

Test the behavior of the `Yoke_male` by dragging it.



16 Add the `spider`.

Use **Insert Component** to add the `spider` component.

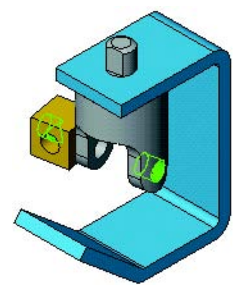


17 Concentric mate for `spider`.

Add a mate between the `spider` and the `Yoke_male`.

Add a **Concentric** mate between the two *cylindrical* faces.

Turn *off* the face **Selection Filter**.



Width Mate

The **Width** mate is the first of the **Advanced Mates** the **Mate** dialog. Selections include a pair of **Width selections** and a pair of **Tab selections**. The **Tab** faces are centered between the **Width** faces to locate the component. The `spider` component should be centered within the `Yoke_male` and `Yoke_female` components.

Note

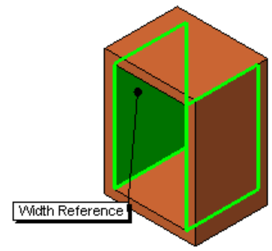
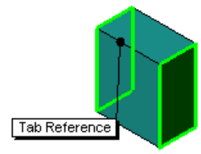
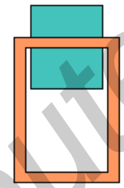
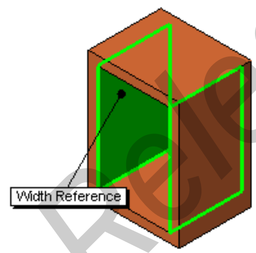
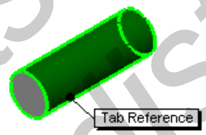
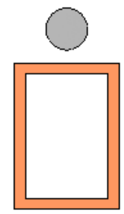
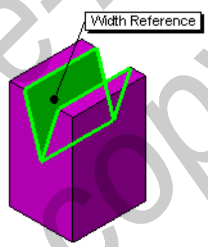
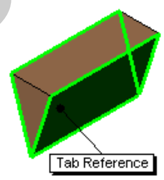
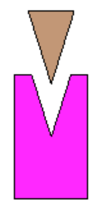
The remaining advanced mates are discussed in the *Advanced Assembly Modeling* manual.

Width References

The **Width** selections form the “outer” faces, used to contain the other component.

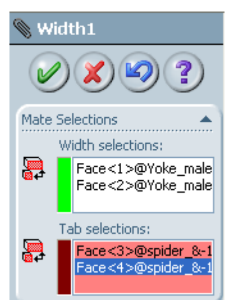
Tab References

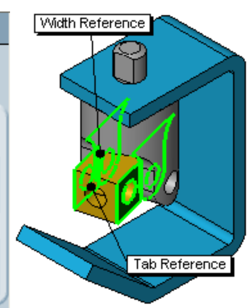
The **Tab** selection(s) form the “inner” faces, used to locate the component.

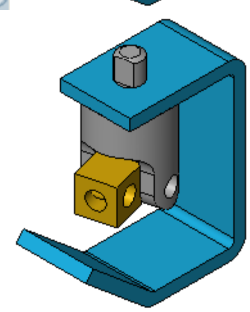
Width selections	Tab selection(s)	Result
		 (Front view)
	 (single selection)	 (Front view)
		 (Front view)

18 Plane to plane mate.
Click **Insert, Mate** and select the **Advanced Mates** tab.

Click the **Width** mate and select the **Width selections** and **Tab selections** as shown.

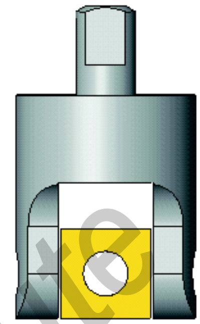






19 Results.

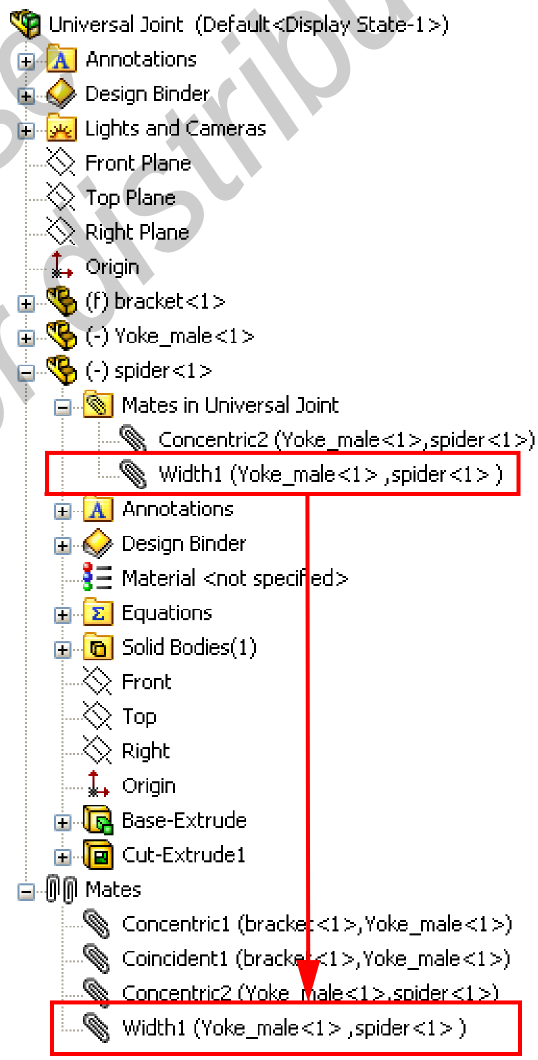
The mate keeps the `spider` centered inside the `Yoke_male` with equal gaps on each side.



20 Mates by component.

Expand the `spider` component in the FeatureManager. A folder named `Mates in Universal Joint` is added to each component that is mated. The folder contains the mates which use geometry of that component.

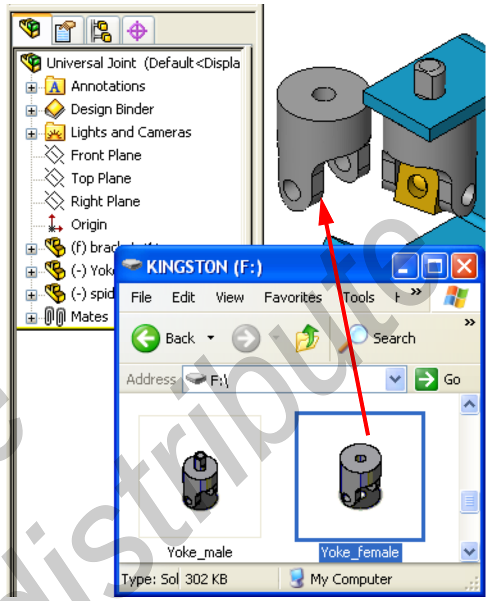
The folder is a subset of the `Mates` folder which contains all mates.



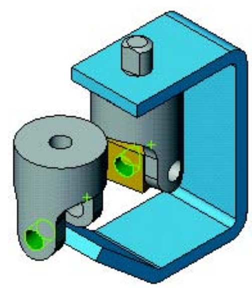
Adding Components Using Windows Explorer

Another way to add components to the assembly is through Windows Explorer or My Computer. The part or assembly file(s) can be dragged and dropped into the active assembly.

21 Open Explorer.
Size the Explorer window so the SolidWorks graphic area can be seen. Since SolidWorks is a native Windows application, it supports standard Windows techniques like "drag and drop". The part files can be dragged from the Explorer window into the assembly to add them. Drag and drop the `Yoke_female` into the graphics area.

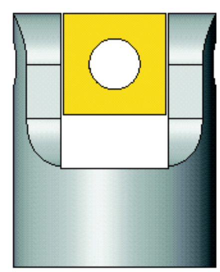


22 Concentric mate.
Select the cylindrical faces as shown and add a **Concentric** mate between them.

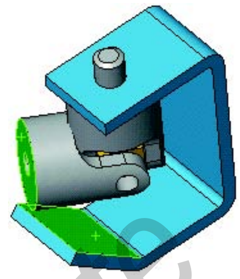


23 Plane to plane mate.
Add a **Width** mate between the `spider` and the `Yoke_female`.

The `spider` is centered on the `Yoke_female` component.



24 Potential over defined condition.
Select the faces of the `Yoke_female` and `bracket` as shown. Because of the clearance between the `Yoke_female` and the `bracket`, a **Coincident** mate is unsolvable. The gap prevents coincidence.



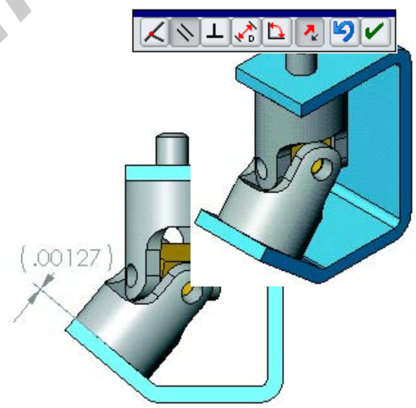
If a **Coincident** mate was selected, a warning dialog would appear:

`Warning: This mate is over defining the assembly. Consider deleting some of the over defining mates.`

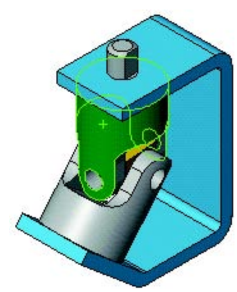
Parallel Mate

A **Parallel** mate keeps the selected planar faces or planes parallel to each other without forcing contact between them.

25 Set to Parallel.
Select the **Parallel** mate to maintain the gap between the faces.



26 Dynamic assembly motion.
Drag any of the under defined components to turn them all.



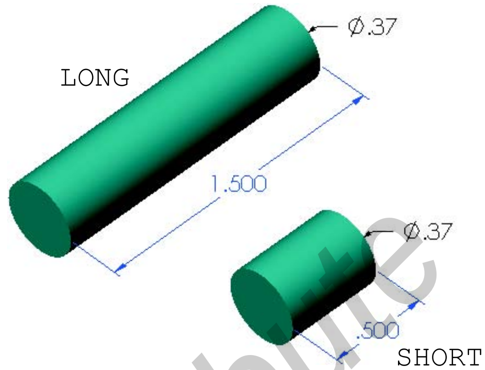
Displaying Part Configurations in an Assembly

When you add a part to an assembly you can choose which of its configurations will be displayed.

Or, once the part is inserted and mated, you can switch its configuration.

The Pin

The part named `pin` has two configurations: `SHORT` and `LONG`. Any configuration can be used in the assembly. In this case, two instances will use `SHORT` and one will use `LONG`.

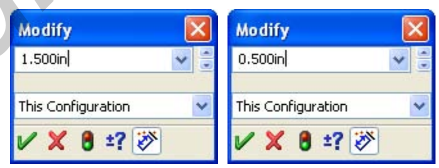


Using Part Configurations in Assemblies

Multiple instances of the same part can be used in an assembly, with each instance referencing a different configuration. We will use multiple instances of a part with different configurations in this assembly.

There are two ways to create this type of configuration within a part:

- Applying different dimension values to individual configurations as shown at the right.
- Design tables.



Drag and Drop from an Open Document

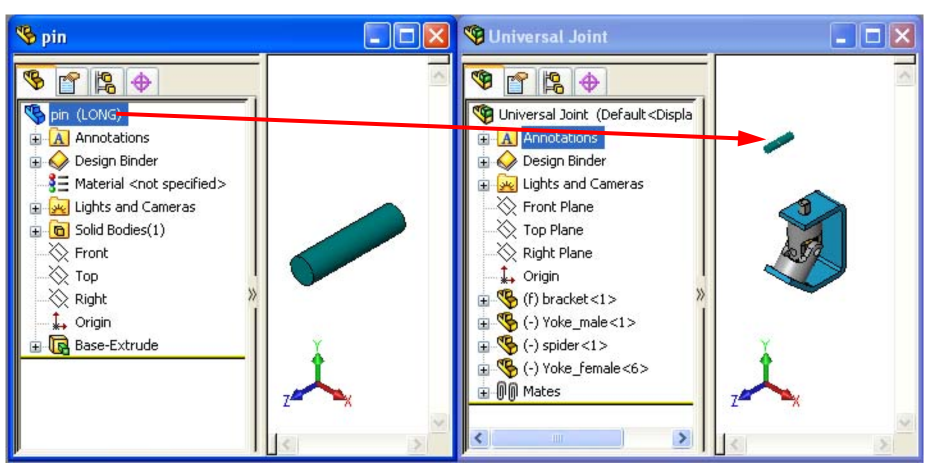
The `pin` will be inserted by dragging it in from an open document window into the assembly.

Note

If the `bracket` window is still open, close it before to the next step.

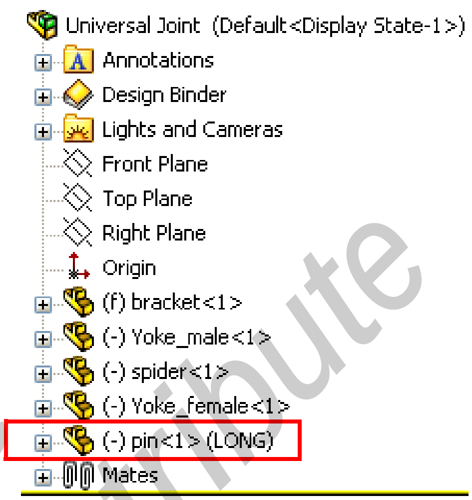
27 Drag and drop.

Open the part `pin` and tile the windows of the assembly and part. Drag and drop the `pin` into the assembly window by dragging the top-level component (pin (LONG)) from the FeatureManager. An instance of the `pin` is added to the assembly.



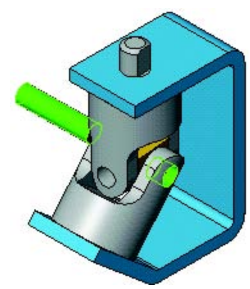
Important!

The `pin` is a component that contains multiple configurations. Components like this display the configuration they are using as part of the component name. In this case the configuration used by instance <1> is `LONG`. Each instance can use a different configuration.

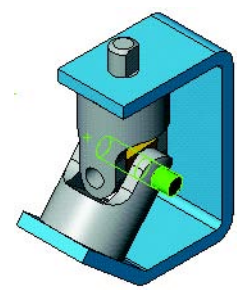


28 Concentric mate.

Add a **Concentric** mate between the cylindrical face in the `Yoke_female` and `pin`.

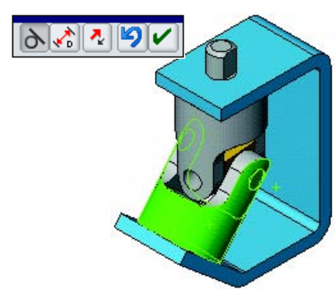


The pin can be dragged while using the mate dialog. Drag it through as shown.



29 Tangent mate.

Add a **Tangent** mate between the planar end face of the `pin` and the cylindrical face in the `Yoke_female`.



The Second Pin

Another instance of the `pin` is needed. This one will be the shorter version, `SHORT`. We will open the `pin`, tile the windows of the part and assembly, and show the part's ConfigurationManager.

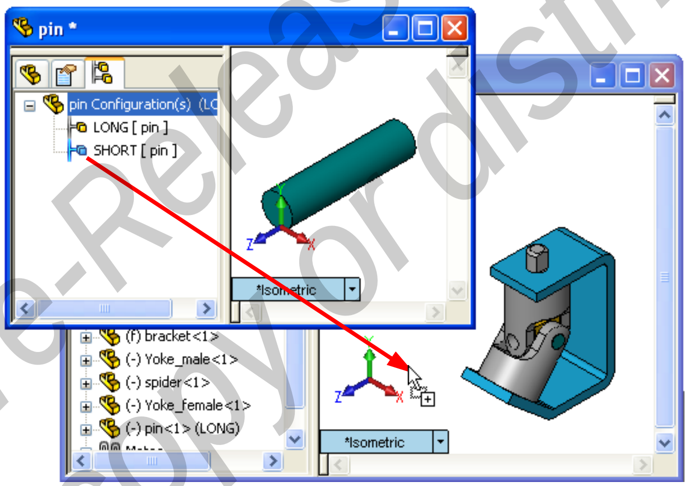
Opening a Component

When you need to access a component while working in an assembly, you can open it directly, without having to use the **File, Open** menu. The component can be either a part or a sub-assembly.

30 Cascade the windows.
Click **Window, Cascade** to see both the part and assembly windows.

Switch to the ConfigurationManager of the `pin`.

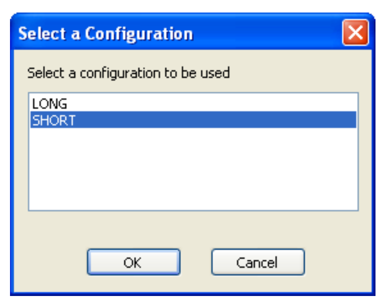
31 Drag and drop a configuration.
Drag and drop the configuration `SHORT` into the graphics window of the assembly. You can drag and drop *any* configuration from the ConfigurationManager, not just the active one.



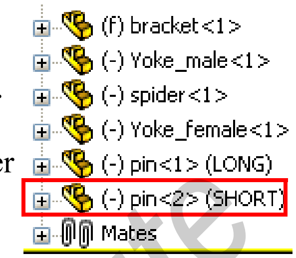
Other Methods of Selecting Configurations

To get the same result using **Insert Component**, browse for the part and associated configuration.

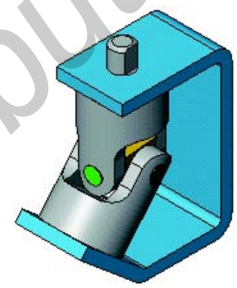
When using Explorer, parts that contain configurations trigger a message box when dragged and dropped. Select the desired configuration from the list.



32 Second instance.
The second instance of the `pin` component is added, this time using the `SHORT` configuration. The component is added and it displays the proper configuration name in the FeatureManager design tree.



33 Mate the component.
Add **Concentric** and **Tangent** mates to mate the second instance of the `pin`.

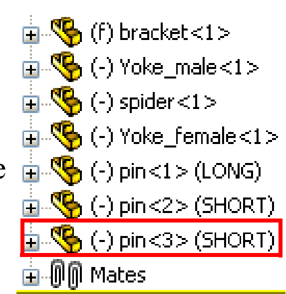


Creating Copies of Instances

Many times parts and sub-assemblies are used more than once in an assembly. To create multiple instances, or copies of the components, copy and paste existing ones into the assembly.

34 Close the `pin` document and maximize the assembly window.

35 Drag a copy.
Create another copy of the `pin` component by holding the **Ctrl** key while dragging the instance with the `SHORT` configuration from the FeatureManager design tree of the assembly. The result is another instance that uses the `SHORT` configuration, since it was copied from a component with that configuration.



You can also drag a copy by selecting the component in the graphics window.

Component Hiding and Transparency

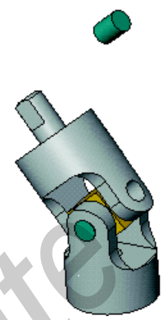
Hiding a component temporarily removes the component's graphics but leaves the component active within the assembly. A hidden component still resides in memory, still has its mates solved, and is still considered in operations like mass property calculations.

Another option is to change the transparency of the component. Selections can be made through the component to others behind it.

**Introducing:
Hide Component
Show Component**

Hide Component turns off the display of a component, making it easier to see other parts of the assembly. When a component is hidden, its icon in the FeatureManager design tree appears in outline form like this: (f) bracket<1>.

Show Component turns the display back on.

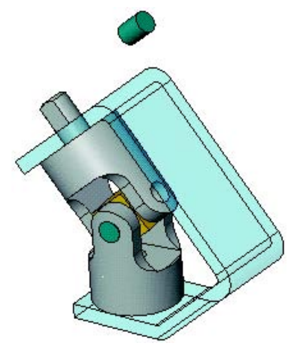


Where to Find It

- Click **Hide/Show Components** on the Assembly toolbar. This acts as a toggle. If the component if visible, it will hide it. If the component is hidden, it will show it.
- Right-click the component and select **Hide** or **Show**.
- Right-click the component and select **Component Properties...** from the **Component** list. Select the **Hide Component** check box.
- From the pull-down menu, choose **Edit, Hide** or **Edit, Show**.

Introducing: Change Transparency

Change Transparency makes the component transparency **75%** and switches it back to **0%**. Selections pass through the transparent component unless the **Shift** key is pressed during selection. The FeatureManager icon does not change when a component is transparent.



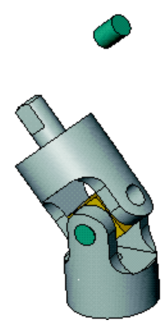
Where to Find It

- Click **Change Transparency** on the Assembly toolbar. This acts as a toggle.
- Right-click the component and select **Change Transparency**.

---

36 Hide the `bracket`.

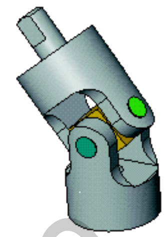
Change the view orientation by pressing **Shift+Left Arrow** once. Click on the `bracket` component and hide it using the **Hide/Show Component** tool. Hiding removes the component's graphics temporarily but leaves the mates intact. The FeatureManager design tree displays the component in *outline* when hidden (f) bracket<1>.



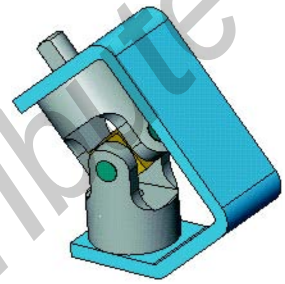
Important!

Use **Hide Component** *not* **Hide Solid Body**. **Hide Solid Body** will hide the solid within the part.

37 Complete the mating.
Complete the mating of this component by adding **Concentric** and **Tangent** mates using **Insert Mate**.



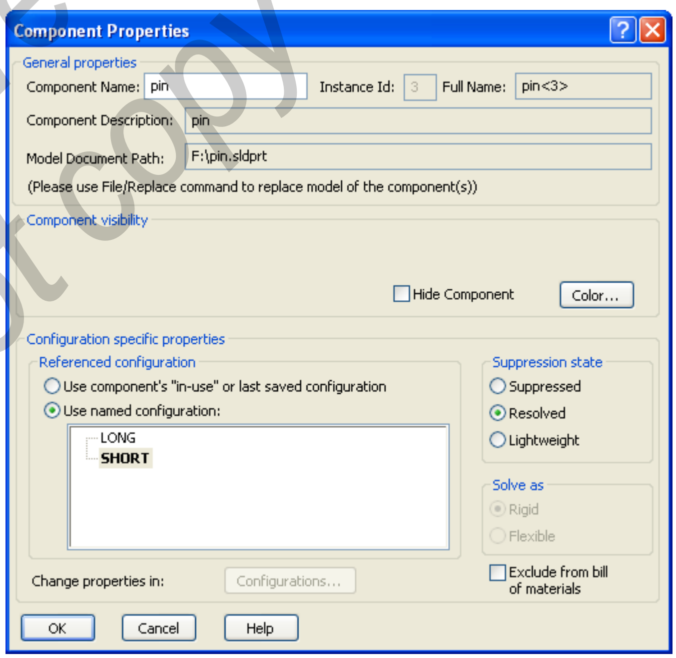
38 Show the component.
Select the bracket again and click **Hide/Show Component** to toggle the graphics back on.



39 Return to previous view.
Previous view states can be recalled using the **Previous View** button on the View toolbar. Each time you press the button, the view display backs up through the display list, whether the view state was saved or not. Click once to return to the previous `Isometric` view.

Component Properties

The **Component Properties** dialog controls several aspects of a component instance.



- **Model Document Path**
  Displays the part file that the instance uses. To replace the file the instance references with a different file, use **File, Replace...**.
- **Visibility**
  Hides or shows the component. Also allows you to change the color of the component *as it appears in the assembly.*

- **Suppression state**
  Suppress, resolve or set the component to lightweight status.
- **Solve as**
  Makes the sub-assembly rigid or flexible. This allows dynamic assembly motion to solve motion at the sub-assembly level.
- **Referenced configuration**
  Determines which configuration of the component is being used.

40 Component properties.
Right-click the pin<3> component and select **Component Properties...** from the **Component** list. The **Use named configuration** option is checked and set to SHORT. This dialog box can be used to change the configuration, suppress, or hide an instance. If **Referenced configuration** is set to **Use component's "in-use" or last saved configuration**, the saved configuration will be displayed.

Click **Cancel**.

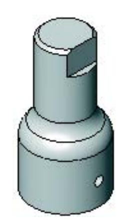
Sub-assemblies

Existing assemblies can also be inserted into the current assembly by dragging. When an assembly file is added to an existing assembly, we refer to it as a sub-assembly. However, to the SolidWorks software, it is still an assembly (*.sldasm) file.

The sub-assembly and all its component parts are added to the FeatureManager design tree. The sub-assembly must be mated to the assembly by one of its component parts or its reference planes. The sub-assembly is treated as a single piece component, regardless of how many components are within it.

A new assembly will be created for the components of the crank. It will be used as a sub-assembly.

41 New assembly.
Create a new assembly using the Assembly_IN template.

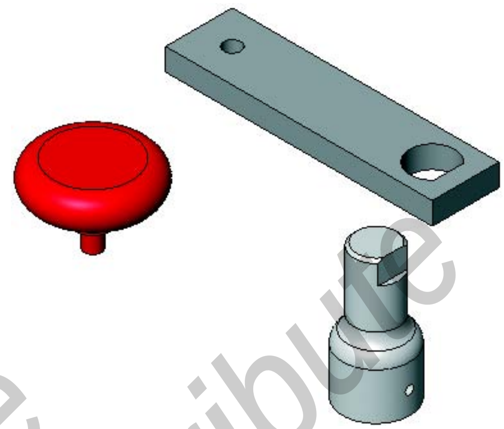


Click **Keep Visible** on the **Insert Component** PropertyManager and add the crank-shaft component. Locate it at the origin of the assembly. It is **Fixed**.

Name the assembly crank sub.

42 Add components.
Using the same dialog, add the `crank-arm` and `crank-knob` components.

Close the dialog.



Smart Mates

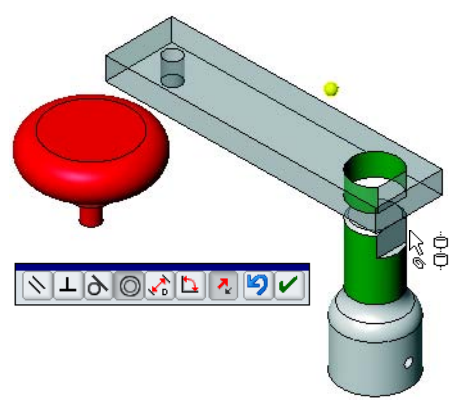
Mates can be added between components while dragging and dropping them. This method, called **Smart Mates**, uses the **Alt** key in conjunction with standard drag and drop techniques.

These mates use the same **Mate Pop-up** Toolbar as the **Mate** tool uses to set the type and other attributes. All mate types can be created with this method.

Certain techniques generate multiple mates and do not use the toolbar. These require the use of the **Tab** key to switch mate alignment.

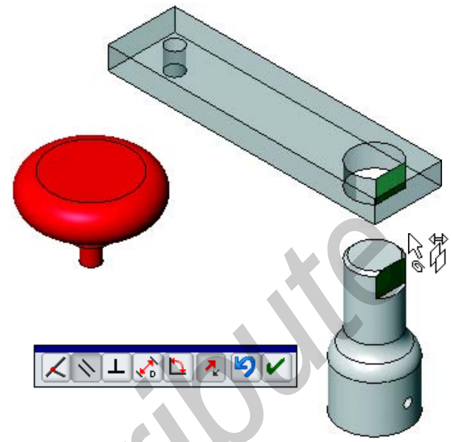
43 Smart Mate concentric.
Follow these steps to add a **Concentric** mate through the **Smart Mate** technique:

1. Click and hold the circular face of the `crank-arm`.
2. Press and hold the **Alt** key as you drag the component.
3. Move the component over the circular face of the `crank-shaft`.
4. Drop the component when the tooltip appears, indicating a concentric mate.
5. Confirm the **Concentric** type from the **Mate Pop-up** Toolbar.



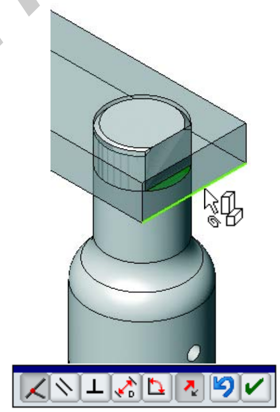
A **Concentric** mate is added between the `crank-arm` and the `crank-shaft` components.

44 Smart Mate parallel.
Spin the `crank-arm` around so the flat is selectable using dragging. Select the flat and **Alt-+drag** it to the flat on the `crank-shaft`. Drop the component when the symbol appears, indicating a **Coincident** mate between planar faces.

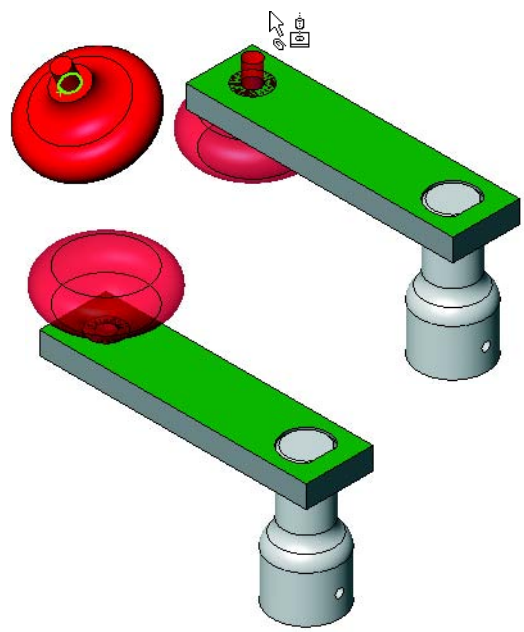


Use the **Mate Pop-up** Toolbar to *switch* to a **Parallel** mate.

45 Coincident.
Select the *edge* of the `crank-arm` and **Alt-+drag** it to the flat on the `crank-shaft`. Drop the component when the symbol appears, indicating a **Coincident** mate between and edge and a planar face. Use the **Mate Pop-up** Toolbar to confirm the **Coincident** mate.



46 "Peg-in-hole".
The "Peg-in-hole" option is a special case of the **Smart Mate** that creates two mates from one drag and drop. This operation is easier if the `crank-knob` has been rotated.



Select the circular edge on the `crank-knob`. Press **Alt** and drag it to the circular edge on the top of the `crank-arm`.

Release the **Alt** key when the symbol appears, indicating that both **Coincident** and **Concentric** mates will be added.

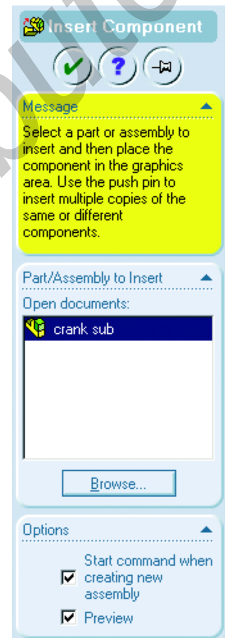
Press the **Tab** key, if necessary, to reverse the alignment. Drop the component.

47 Save.
Save the assembly but leave it open.

Inserting Sub-assemblies

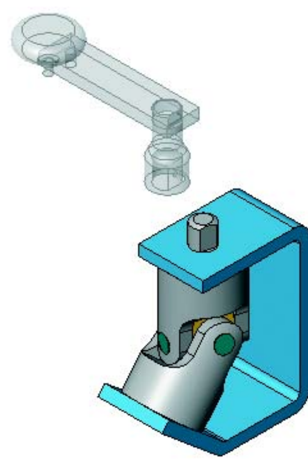
Sub-assemblies are existing assemblies that are added to the active assembly. All of the components and mates act as a single component.

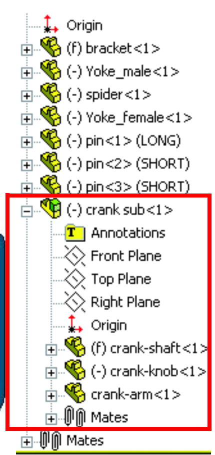
48 Select the sub-assembly.
Using **Insert Component**, the dialog is set to list any open parts or assemblies under **Open documents**. The `crank sub` is listed and selected.



49 Place the sub-assembly.
Place the sub-assembly near the top of the `Yoke_male` component.

Expanding the sub-assembly component icon shows all the component parts within it, including its own mate group.

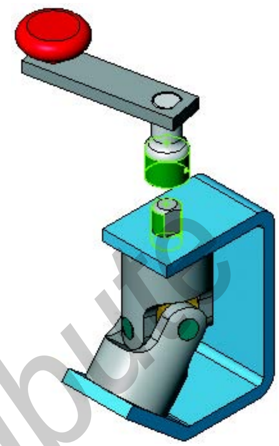




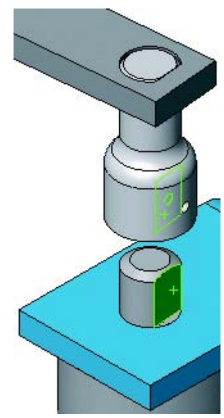
Mating Sub-assemblies

Sub-assemblies follow the same rules for mating as parts. They are considered components and can be mated using the **Mate** tool, **Alt+drag** mating or a combination of both.

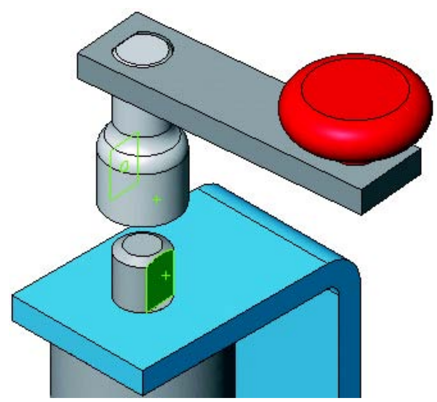
50 Smart Mate concentric.
Add a **Concentric** mate, using **Alt+drag**, between the cylindrical surfaces of the post on the top of the `Yoke_male` and the `crank-shaft`.



51 Parallel mate.
Mate the flat on the `Yoke_male` with the flat in the `D-hole` in the `crank-shaft` using the **Mate** tool and a **Parallel** mate.



52 Alignment.
Click the **Flip Mate Alignment** button to test **Anti-Aligned** (above) and **Aligned** (right). Use the anti-aligned condition for this mate.



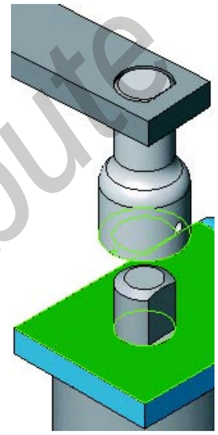
Question: Why wouldn't you use a **Coincident** mate here?

Answer: Because unless the dimensions of the flats and the diameters of the shaft and corresponding hole are exactly right, a coincident mate would over define the assembly.

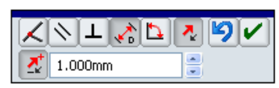
Distance Mates

Distance mates allow for gaps between mating components. You can think of it as a parallel mate with an offset distance. There is generally more than one solution so the options **Flip Mate Alignment** and **Flip Dimension** are used to determine how the distance is measured and what the side it is on.

53 Select the faces.
Select the top face of the `bracket` and the bottom face of the `crank-shaft` component to create the mate.



54 Add a Distance mate.
Specify a distance of **1mm**.



Note

Even though the units of this assembly and all of its components are inches, you can enter metric values in the spin boxes. Just type **mm** after the number. The system will automatically convert it to 0.039 inches.

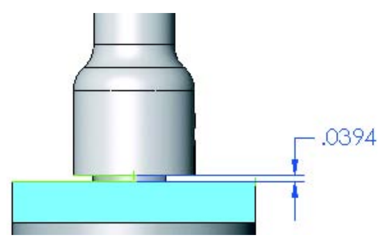
Click **Preview**.

If the `crank-shaft` penetrates into the `bracket` select the **Flip Dimension** button.

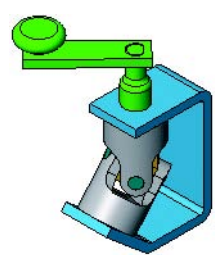
Click **OK** to create the mate.

Tip

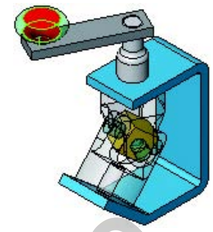
Double-clicking a **Distance** or **Angle** mate in the FeatureManager displays it on the screen. The value displays in the units of the assembly, in this case inches.



55 Select in the FeatureManager.
Select the sub-assembly `crank sub` in the FeatureManager design tree. All components in the sub-assembly will be selected and highlighted light green.



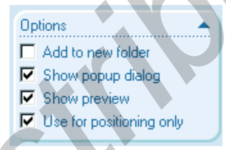
56 Dynamic Assembly Motion.
Use **Change Transparency** on the yokes and pins. Move the handle to see the motion of the `spider`.

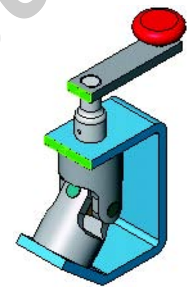


Use For Positioning Only

The mate option **Use for positioning only** can be used to position geometry without adding the restriction of a mate. This is a useful method for setting up a drawing view.

57 Mate.
Click the **Mate** tool and select **Use for positioning only**. Select the planar faces shown and a **Parallel** mate. Click **OK**.



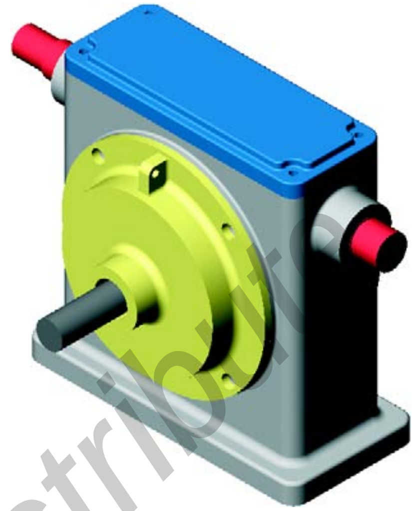


The geometry is positioned like a parallel mate condition but no mate is added.

58 Save and close.

Exercise 53: Gearbox Assembly

Create this assembly by using mates only. No dimensions are provided.



This lab uses the following skills:

- Bottom-Up assembly creation.
- Adding components to an assembly.
- Creating mates between components using **Insert Mate**.

Units: **inches**

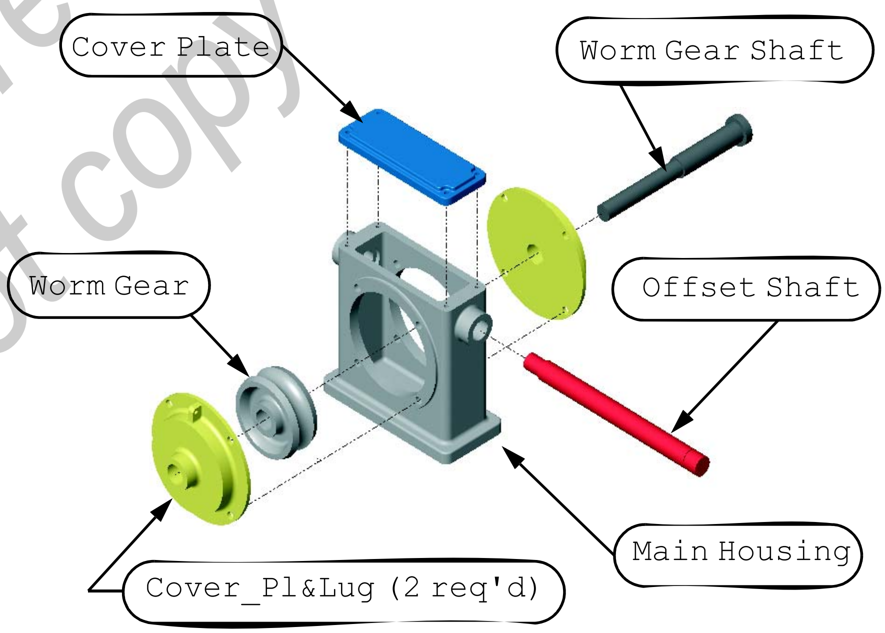
Design Intent

The design intent for this part is as follows:

1. Files are found in the `Gearbox Assy` folder.
2. Component parts are mated as shown in the details.
3. Two instances of the `Cover_Pl&Lug` are required.

Part Design

Use the following graphics along with the design intent to determine the shape and relationships within the assembly.



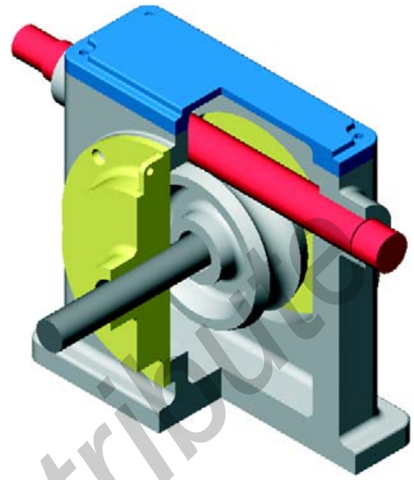
Procedure

Open a new assembly using the `Assembly_IN` template.

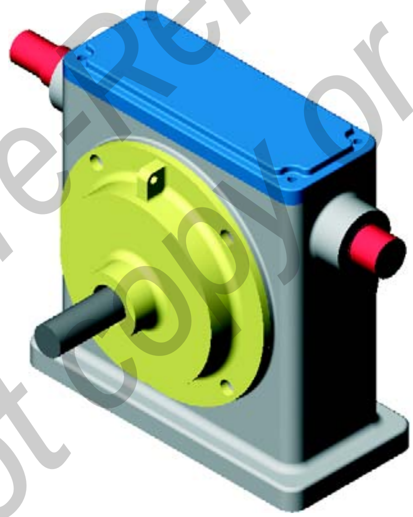
1 Add the components.
Drag the component parts into a new assembly document.

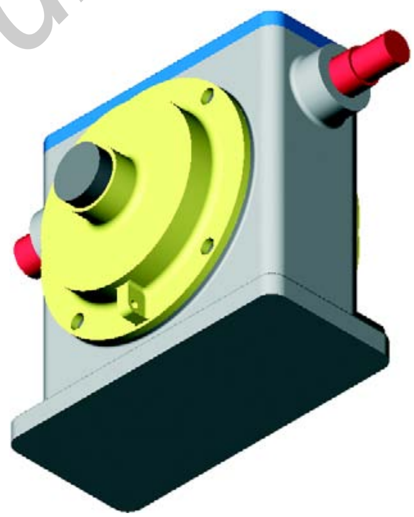
2 Mate the components.
Mate the `Housing` to the origin of the assembly. Mate the other components to the `Housing` and each other.

3 Cutaway of the assembly.
This cutaway shows the internal components of the assembly. Use this and the following details to mate components in the assembly.



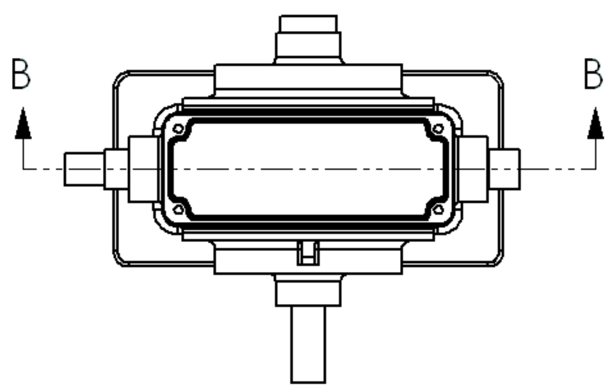
4 Orientation of components.
The `Cover_Pl&Lug` components are oriented differently on the front and back. Note the position of the lug in these views.



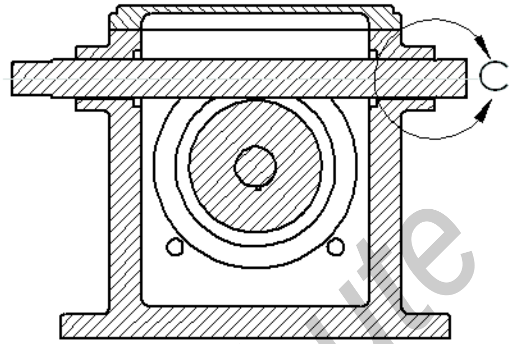


Note You do not need to create these section views. They are provided for information purposes only.

- **`Top` view.**
This view shows the cutting line for `Section B-B`.

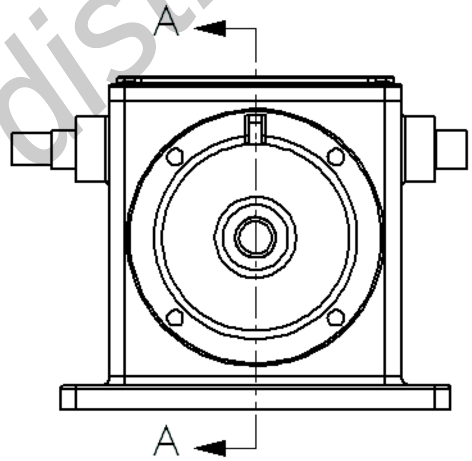


- **Section B-B.**
  Section B-B is cut from the Top view.

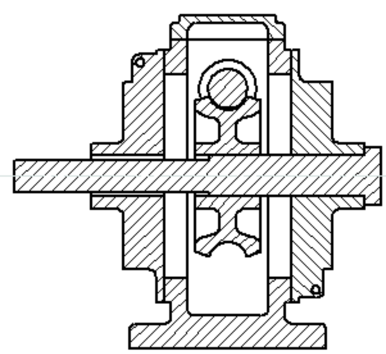


SECTION B-B

- **Front view.**
  This view includes the cutting line for Section A-A.



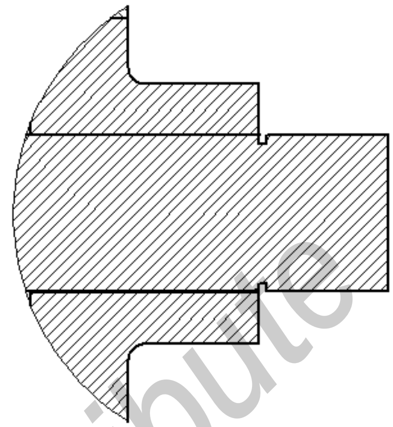
- **Section A-A.**
  Section A-A is cut from the Front view.



SECTION A-A

- **`Detail C`.**
  `Detail C` is taken from `Section B-B` and includes the mating of the groove.

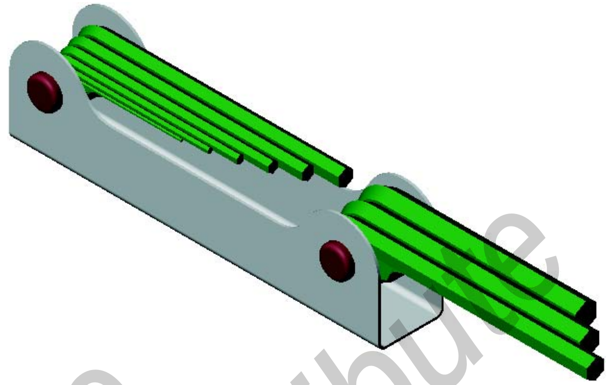
5 Save and close the assembly.



DETAIL C
SCALE 2 : 1

Exercise 54: Part Design Tables in an Assembly

Using the parts included, complete this bottom up assembly. Use several configurations of the same part in the assembly to create a set of allen wrenches.



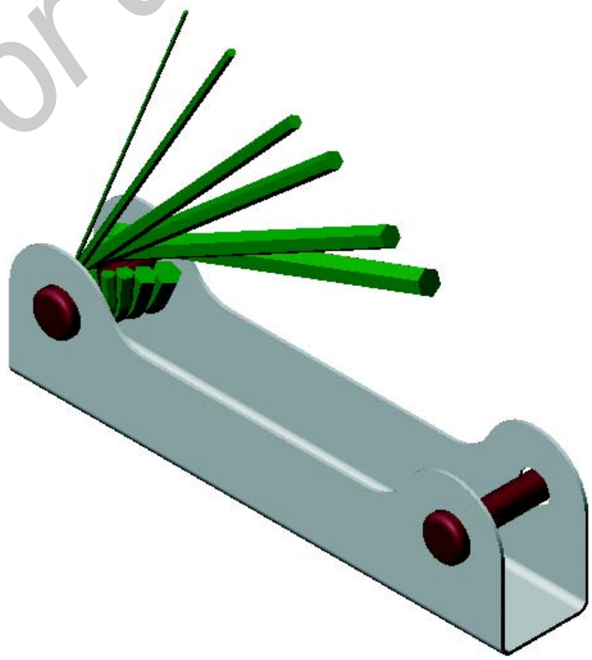
This lab reinforces the following skills:

- Configurations in a part.
- Part configurations in assemblies.
- Editing design tables.
- Bottom-up assembly design.
- Exploded views and explode lines.

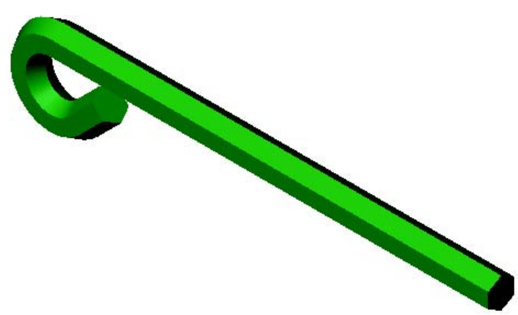
Procedure

Open an existing assembly.

1 **Existing assembly.**
Open the existing assembly named `part configs`. It is located in the folder named `Part DT in Assy`. The assembly contains three components, two of which have multiple instances. One component, the `Allen Wrench`, uses a different configuration for each instance.



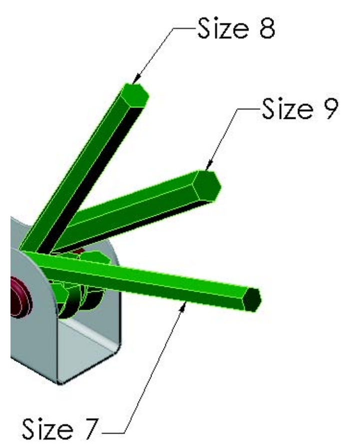
2 **Open part.**
Select any instance of the `Allen Wrench` component and open the part.



3 Design table.
Edit the design table that is embedded there. Change the values in the `Length@Sketch1` column only.

	`Length@Sketch1`
Size01	50
Size02	60
Size03	70
Size04	80
Size05	90
Size06	100
Size07	100
Size08	90
Size09	80
Size10	100

4 Add and mate components.
Add and mate three more components, noting the configurations of the `Allen Wrench` parts. The sizes, positions and part names are detailed in the accompanying illustrations.



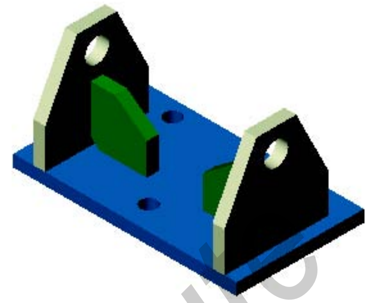
Hint

With the part and the assembly both open, tile the windows. Switch to the ConfigurationManager in the part and drag in only the configurations that you need.

5 Save and close the assembly and the part.

Exercise 55: Mates

Create this assembly by adding components to a new assembly and using **Insert Mate**.



This lab uses the following skills:

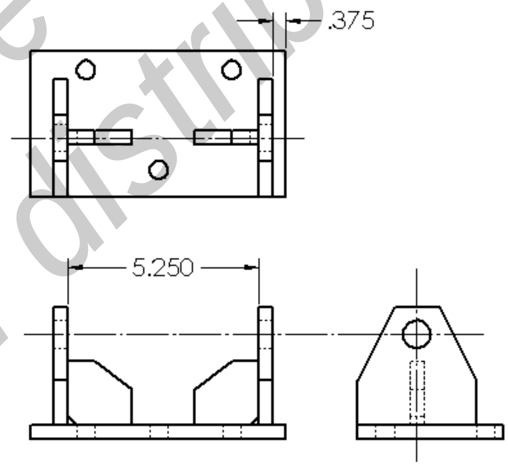
- Bottom-Up assembly creation.
- Adding components to an assembly.
- Creating mates between components using **Insert Mate**.

Units: **inches**

Design Intent

The design intent for this part is as follows:

1. Files are found in the `Mates` folder.
2. Component parts are mated as shown in the details.
3. Two instances of the `Brace` and `EndConnect` are required.
4. Each `Brace` component is centered on the hole in the `EndConnect` component.

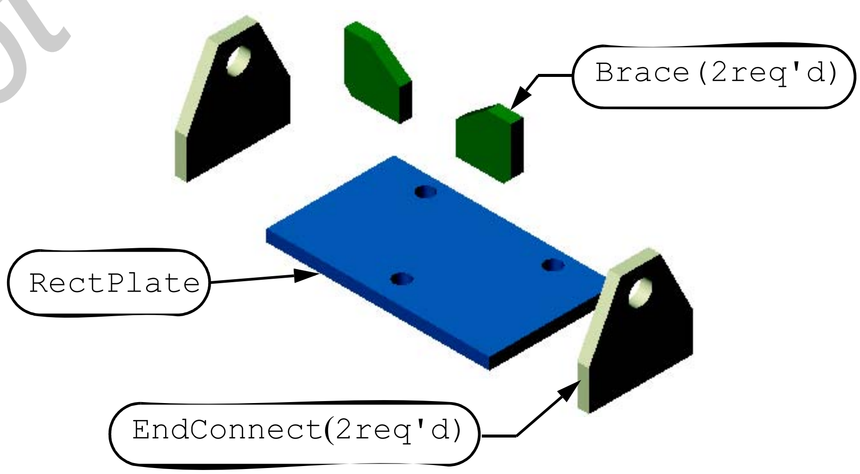


Tip

Mates between planes can be used to center components.

Part Design

Use the following graphics along with the design intent to determine the shape and relationships within the assembly.

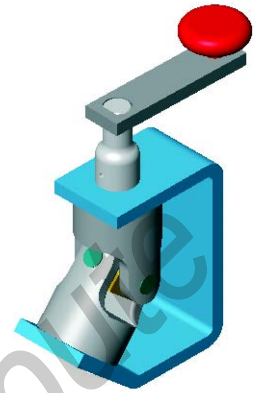


Exercise 56: U-Joint Changes

Make changes to the assembly created in the previous lesson.

This exercises uses the following skills:

- Opening parts from the assembly.
- Changing part dimensions.
- Adding and deleting mates.
- Adding components.

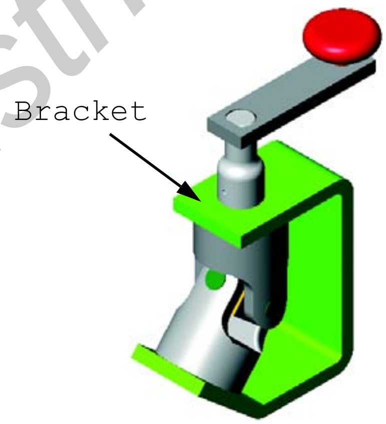


Procedure

Open an existing assembly.

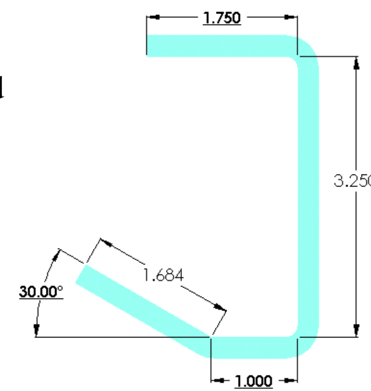
1 **Open the assembly named `Changes`.**
The assembly is found in the `U-Joint Changes` folder.

2 **Open the `bracket` component.**
From the FeatureManager or the screen, open the component `bracket<1>` for editing.



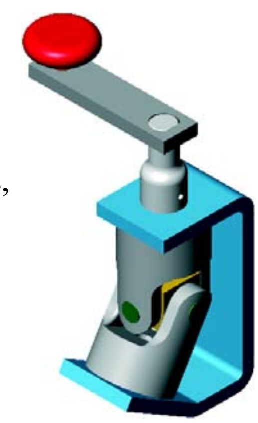
3 **Changes.**
Double-click the first feature and change the dimensions that are shown as bold and underlined.

Rebuild the part.



4 **Close and save.**
Close the `bracket` part saving the changes that you have made. Respond **Yes** to rebuilding the assembly.

5 **Changes.**
The changes made in the part also appear in the assembly.



6 **Turn the crank.**
The crank should turn freely, turning the two yokes, the spider, and the pins with it.

7 **Delete mate.**
Expand the mate group and delete the mate `Parallel2`.

8 Turn the crank.

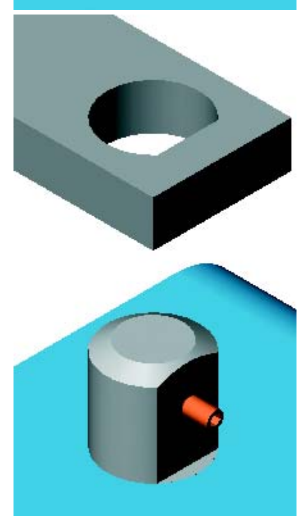
The crank should turn freely but it is no longer connected to the yokes and spider.

9 Insert a `set screw`.

Insert the existing component named `set screw`. Mate it to the small hole in the `crank-shaft` with a **Concentric** mate.

10 Hide component.

Hide the `crank-shaft` component. Add a **Coincident** mate between the flat faces of the `set screw` and the `Yoke_Male`.



11 Show component.

Show the `crank-shaft` component.

12 Turn the crank.

The crank should turn freely and once again, the two `yokes`, the `spider` and the `pins` should rotate with it.

13 Save and close the assembly.

Exercise 57: Gripe Grinder

Assemble this device by following the steps as shown.

This lab uses the following skills:

- Bottom-up assembly modeling.
- Dynamic assembly motion.
- Configuration of parts in an assembly.

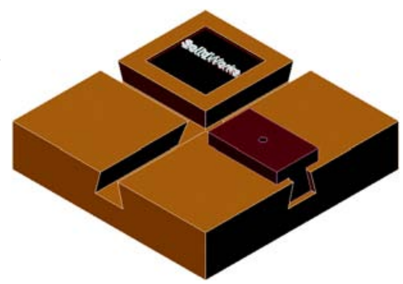
Procedure

Open a new assembly using the `Assembly_IN` template.

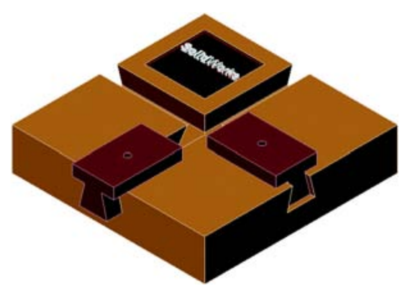
1 Add the component `Base`.
The parts for this assembly are in the folder named `Grinder Assy`.

Drag the `Base` into the assembly and fully constrain it to the assembly origin.

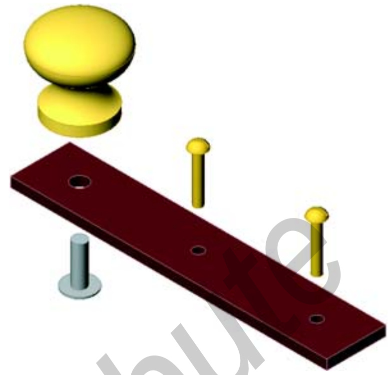
2 Add the `Slider`.
Add the `Slider` to the assembly. **Mate** it to one of the dovetail slots. A width and coincident mate are required.

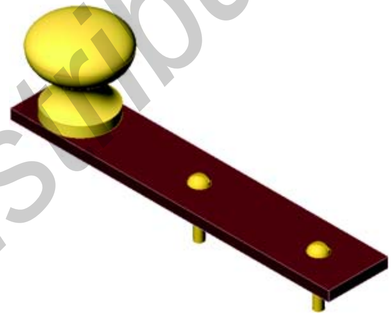


3 Add a second copy of the `Slider`.
Mate it to the other dovetail slot. Both `Sliders` should be free to move back and forth in their respective slots.



4 `Crank` assembly.
Open a new assembly using the `Assembly_IN` template. Build the `Crank` assembly as shown at the right. The `Crank` is shown in both exploded and collapsed states.



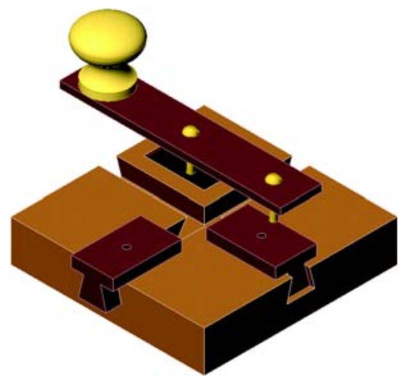


The `Crank` assembly consists of:

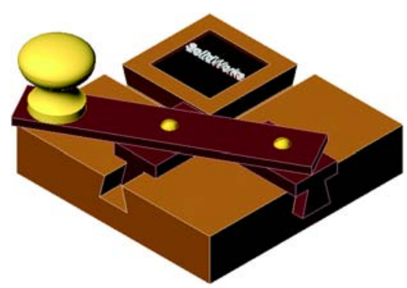
- `Handle` (1)
- `Knob` (1)
- `Truss Head Screw` (1) [#8-32 (.5" long)] configuration
- `RH Machine Screw` (2) [#4-40 (.625" long)] configuration

Note Both machine screws contain multiple configurations. Be sure you use the correct ones.

5 Insert the `Crank` assembly into the main assembly.
Tile the two assembly windows, and drag and drop the sub-assembly into the main assembly.



6 Mate the `Crank` assembly to the `Sliders`.
The two `RH Machine Screws` go into the holes in the `Sliders`. The underside of the `Handle` mates to the top face of one of the `Sliders`.



7 Turn the `Crank`.
The movement of the `Knob` follows an elliptical path. The movement of each `Slider` traces the major and minor axes of that ellipse.

Lesson 13
Using Assemblies

Upon successful completion of this lesson, you will be able to:

- Perform mass properties calculations and interference detection.
- Create an exploded view of an assembly.
- Add explode lines.
- Generate a Bill of Materials for an assembly.

Using Assemblies

This lesson will examine other aspects of assembly modeling using a version of the universal joint assembly. The completed assembly will be analyzed, edited and shown in an exploded state.

Stages in the Process

Some key stages in the analysis process of this part are shown in the following list. Each of these topics comprises a section in the lesson.

- **Analyzing the assembly**
  You can perform mass properties calculations on entire assemblies. You can also perform static or dynamic interference detection.
- **Editing the assembly**
  Individual parts can be edited while in the assembly. This means you can make changes to the values of a part's dimensions while active in the assembly.
- **Exploded assemblies**
  Exploded views of the assembly can be created by selecting the components and the direction/distance of movement.
- **Bill of Materials**
  A BOM table can be generated from the assembly and placed on the drawing sheet. Associated balloons can be added to identify the items.

Analyzing the Assembly

There are several types of analysis you can perform on an assembly. These include calculating the mass properties of the assembly and checking for interferences.

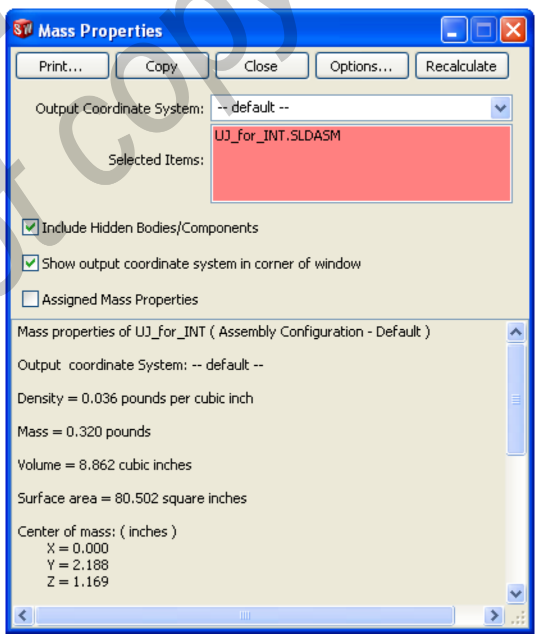
Mass Properties Calculations

Mass properties calculations were introduced in *Lesson 6: Revolved Features*. When working with assemblies, the important thing to remember is that the material properties of each component are controlled individually via the Material feature in the part. The material properties can also be set through **Edit Material**.

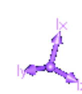
To review material properties, see *Edit Material* on page 193. To review mass properties calculations, see *Mass Properties* on page 195.

1. **Open existing assembly.**
   Open the existing assembly `UJ_for_INT`.
2. **Mass properties.**
   Click **Mass Properties** on Tools toolbar.
3. **Results.**
   The system performs the calculations and displays the results in a report window. The system also displays the **Principal Axes** as temporary graphics. **Options** can be used to change the units of the calculations.

   Click **Close**.



The symbols represent:

Output coordinate system		**Principle Axes** at the **Center of Mass**	

Checking for Interference

Finding interferences between *static* components in the assembly is the job of **Interference Detection**. This option takes a list of components and finds interferences between them. The interferences are listed by paired components including a graphic representation of the interference. Individual interferences can be ignored.

Introducing: Interference Detection

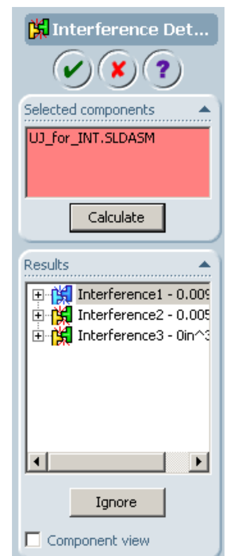
Interference Detection is used to find interferences (clashes) between component parts in an assembly. It can be directed to check all components in the assembly, or just selected ones.

Where to Find It

- Click **Interference Detection** on the Assembly toolbar.
- From the **Tools** menu choose: **Interference Detection...**.

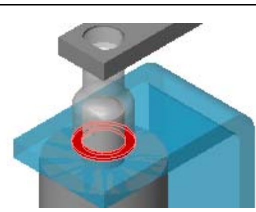
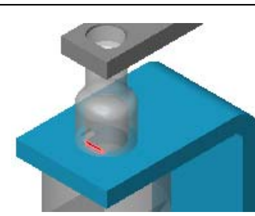
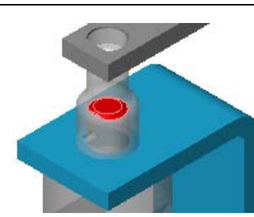
4 **Click Tools, Interference Detection....**
The **Interference Detection** PropertyManager opens.

5 **Interference detection.**
Select the top level component `UJ_for_INT` to check all the components in the assembly. The assembly `UJ_for_INT.SLDASM` appears in the **Selected Components** list.



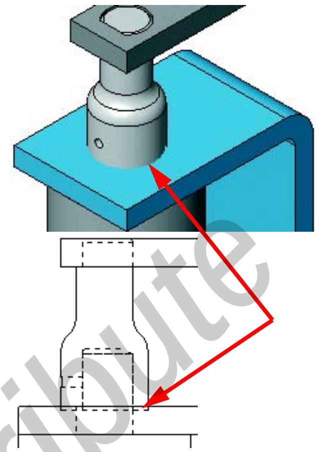
Click **Calculate**.

6 **Interferences.**
The analysis has found three interferences among the selected entities. The listings `Interference1`, `Interference2` and `Interference3` are shown in the **Results** listing followed by a volume of interference. The interference is marked in the graphics window using a volume displayed in red. By default, the interfering components are transparent and the other components remain opaque. Click **OK**.

Interference1	Interference2	Interference3
		
`bracket`	`Yoke_male`	`Yoke_male`
`crank-shaft`	`crank-shaft`	`crank-shaft`

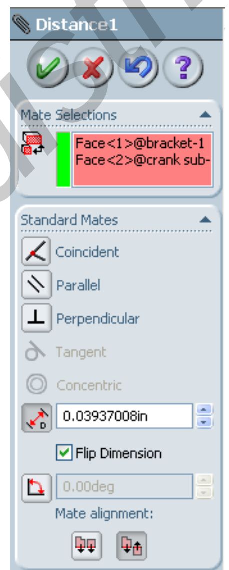
7 Visual methods.
Areas of interference can sometimes be determined visually. **Shaded** (without edges) and **Hidden Lines Visible** displays can be used.

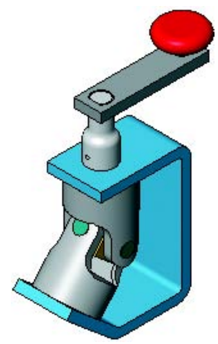
In this case, the `crank-shaft` volume overlaps that of the `bracket`.



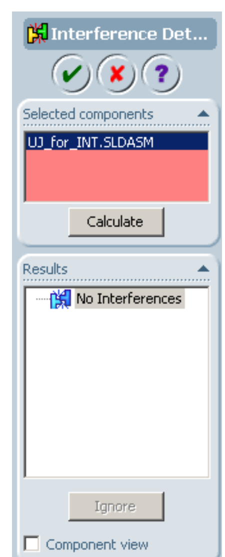
8 Edit Feature.
Right-click the `Distance1` mate and choose **Edit Feature**.

Click the **Flip Dimension** option and click **OK**.





9 Recheck the interferences.
Select the `bracket`, `crank-shaft` and `Yoke_male` components and click **Interference Detection**. As expected, `No Interference` is the result.



Static vs. Dynamic Interference Detection

The problem with a static method of interference detection is that the components of an assembly may only interfere under certain conditions. What is needed is a way to detect collisions dynamically, while an assembly is moving.

Introducing: Collision Detection

Collision Detection analyzes selected components in the assembly during dynamic assembly motion, alerting you when faces clash or collide. You have the options of stopping the motion upon collision, highlighting the colliding faces, and generating a system sound.

Where to Find It

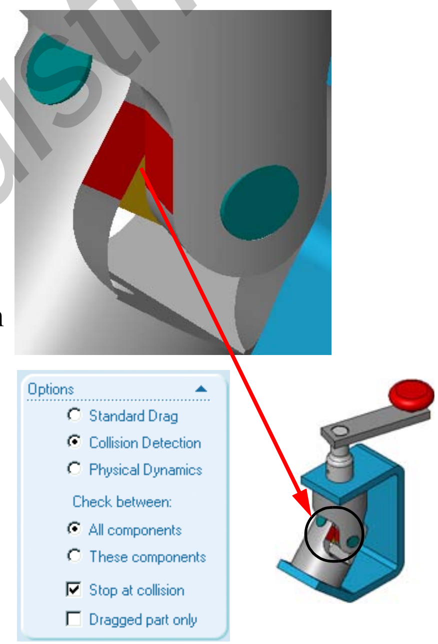
- On the **Move Component** or **Rotate Component** PropertyManagers, select **Collision Detection**.

10 Collision Detection.

Click **Move Component** and check **Collision Detection**.

Check **All components** and **Stop at collision**.

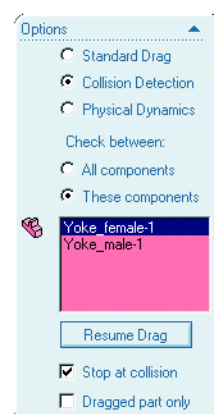
Turn the U-joint by dragging the crank handle. When the inner edges of the two `yokes` collide, the system alerts you by highlighting the faces and generating a system sound.



11 Narrow the selection.
The option **All components** means collisions with *all* assembly components are detected. This puts more demands on system resources, especially in a large assembly. If you choose **These components**, only collisions with a group of assembly components that you select are detected.

Click **These components** and select the `Yoke_female` and `Yoke_male` components.

Click **Stop at collision** and then **Resume Drag**.



12 Turn off Collision Detection.
Click **OK** to close the PropertyManager.

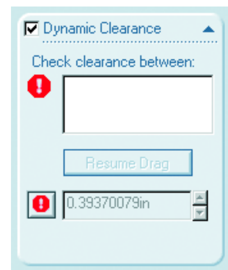
Performance Considerations

The are a number of options and techniques you can use to improve system performance during **Dynamic Collision Detection**:

- Click **These components**, instead of **All components**. In general, performance can be improved if you minimize the number of components the system has to evaluate. However, be careful that you do not overlook a component that does, in fact, interfere.
- Make sure **Dragged part only** is selected. This means only collisions with the component you are dragging are detected. If unchecked, collisions are detected for both the moving component and any components that move as a result of mates to the moving component.
- If possible, use **Ignore complex surfaces**.

Note

The **Dynamic Clearance** option can be used to display the actual clearance between components as they move. A dimension appears between the selected components, updating as the minimum distance between them changes.

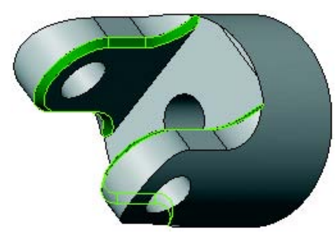


Correcting the Interference

Filleting or chamfering the edges of the yokes will eliminate the interference.

13 Open part.
In the FeatureManager design tree, right-click the `Yoke_female` and select **Open Part**.

Add a **0.05” x 45 chamfer** to the edges as shown. Save the changes.



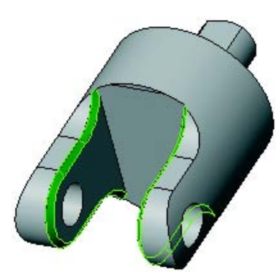
14 Return to the assembly.
Click **Window, `UJ_for_INT.SLDASM`** or by using **Ctrl+Tab**.

When the software detects the change in the part, you will be prompted with a message asking if you would like to rebuild the assembly.

Click **No** in response to the message until all changes have been made.

15 Correct the `Yoke_male` component.
Open the `Yoke_male` using **Open Part**. Add a chamfer the same way as was done in the `Yoke_female` component.

Save the changes and return to the assembly, clicking **Yes** on the **Rebuild Assembly** message.



16 Check for interference.
Click **Move Component**. Click these options:

- **Collision Detection**
- **All components**
- **Stop at collision**

Test for interference by turning the crank. No collisions are detected.

17 Turn off the Move Component tool.

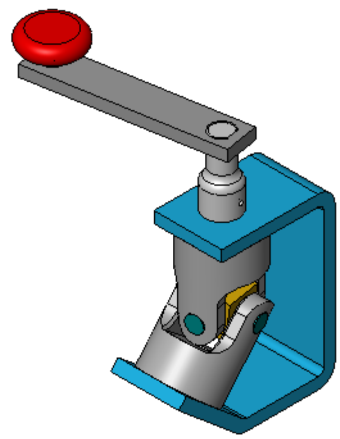
Changing the Values of Dimensions

Changing the value of a dimension in the assembly works exactly the same as changing that dimension in a part: double-click the feature and then double-click the dimension. SolidWorks uses the same part in the assembly or the drawing, so changing it in one place changes it in all.

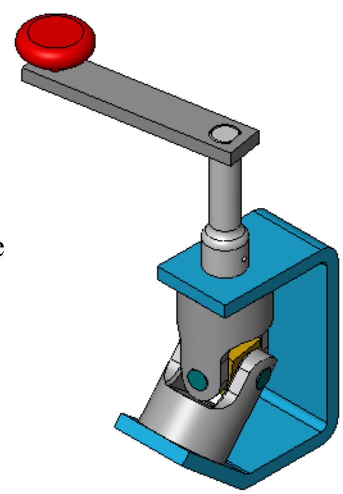
The feature can be double-clicked from the FeatureManager or the screen, but the dimension will always appear on the screen.

18 Edit the `crank-arm`.
Double-click on the graphics of the `crank-arm` part to access its dimensions. These are the dimensions used to build the part.

Change the length to **4”**.

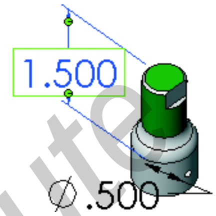


19 Edit the `crank-shaft`.
Change the value of the length to **2.5”**. Notice that not only are the parts rebuilt and the assembly updated, the mating relationships ensure that the `crank-arm` moves up when the `crank-shaft` gets taller and the `crank-knob` moves when the `crank-arm` gets longer.



20 Open `crank-shaft`.
Right-click the `crank-shaft` and select **Open Part** from the shortcut menu.

21 Part level changes.
Changing a part at the assembly level changes it at the part level and vice-versa. That is because it is the same part, not a copy.



Change the value back to **1.5”** and close the part, saving the changes.

22 Assembly update.
Changes have been made to a reference of the assembly, in this case the size of a part. Upon reentering the assembly, SolidWorks asks whether you want to rebuild. Click **Yes**.

23 Change values back.
Select and change the dimension of the `crank-arm` back to **3”** and rebuild.

Using Physical Dynamics

Physical Dynamics is a method for visualizing assembly motion in a more realistic way. Expanding on the capabilities of dynamic collision detection, **Physical Dynamics** lets one object act upon another. When two objects collide, one will move the other according to the available degrees of freedom. **Physical Dynamics** propagates throughout the assembly. The dragged component can push aside a component, which then moves into and pushes aside another component, and so on.

Note

Do not confuse **Physical Dynamics** with a kinematic analysis application such as COSMOSMotion. With **Physical Dynamics**, characteristics such as momentum, friction, or whether a collision is elastic or inelastic are not considered.

Where to Find It

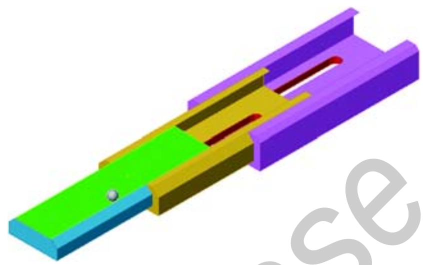
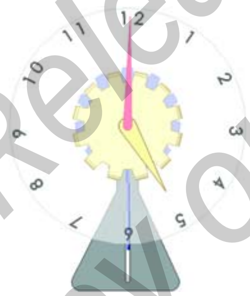
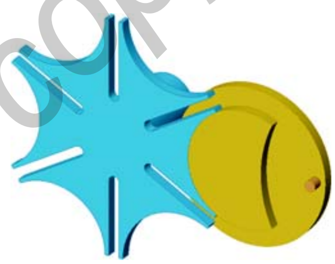
- On the **Move Component** PropertyManager, click **Physical Dynamics**.

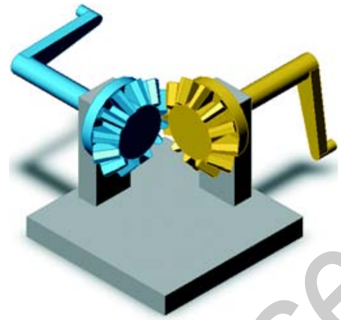
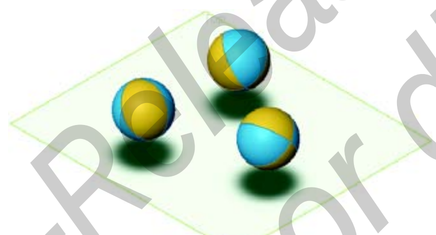
What is this Thing?

When you drag a component with **Physical Dynamics** enabled, at small symbol ⊕ appears on the component. This represents the center of mass. **Physical Dynamics** uses mass properties to compute how the forces acting on a component will make it behave as it collides with other components. Dragging a component by its center of mass exhibits different motion than dragging by a point on the component.

Examples

In the `Physical Dynamics` folder are some examples. They are illustrated in the chart below.

Simulation Element	Description
`Nested Slides` 	As you drag the innermost slide, the next slide is contacted and pulled out as far as possible.
`Clock` 	As you drag the minute hand, the hour hand moves.
`Geneva Wheel` 	As you turn the input wheel, the pin engages and disengages the slots in the output wheel.
`Limit Mechanism` 	Rotate the cam wheel counter-clockwise and the Y-shaped actuating lever oscillates back and forth.

Simulation Element	Description
Bevel Gears 	Turn the handle on one gear, and the other gear rotates.
Rolling Balls 	Drag the individual balls so they collide with each other.

Tips for Working With Physical Dynamics

There are some things you should keep in mind when you use **Physical Dynamics**.

1. **Physical Dynamics** depends on collision detection. It will not work if the assembly contains interferences. If the item you are dragging interferes with another component, the source of the interference is made transparent. Use **Tools, Interference Detection** to find and eliminate interferences before using **Physical Dynamics**.
2. Use the appropriate mates to define the assembly. Highly unconstrained assemblies are less likely to be successful. Do not depend on **Physical Dynamics** to solve everything. For example, in the `Nested Slides` assembly, the appropriate mates were used to mate `slide1` and `slide2` so they each had only one degree of freedom. Then **Physical Dynamics** was used to handle the interaction of the pins and the slots.
3. **Physical Dynamics** does not work on assemblies that have symmetry mates. For more information about symmetry mates, see the advanced course *Advanced Assembly Modeling*.
4. **Physical Dynamics** can be computationally intensive. Limit the scope by selecting components in the **Selected Items** box, and then clicking **Resume Drag**. Items that are not in the list are ignored.

Physical Simulation

Physical Simulation allows you to simulate the effects of motors, springs, and gravity on your assemblies. **Physical Simulation** combines simulation elements with SolidWorks tools such as mates and Physical Dynamics to move components around your assembly. Use an assembly that has the mates to support the simulation effects.

Note

Do not confuse **Physical Simulation** with a kinematic analysis application. With **Physical Simulation**, characteristics such as momentum, friction, or whether a collision is elastic or inelastic are not considered.

Simulation Toolbar

The commands for **Physical Simulation** are located on the Simulation toolbar. The individual tools will be explained latter in this lesson.

Where to Find It

- Click **Simulation Toolbar** on the Assembly toolbar.
- Or, click **View, Toolbars** and select **Simulation**.

Toolbar Options

There are several options for creating the simulation:

Stop Record or Playback | **Calculate Simulation**
Reset Components | **Replay Simulation**

Simulation Elements

There are several simulation elements that move components around in the assembly.

Simulation Element	Description
Linear Motor	Linear Motors move components along a straight line path.
Rotary Motor	Rotary Motors move components about a selected axis, but they are not forces. Motor strength does not vary based on component size or mass. For example, a small component moves at the same speed as a large component if the **Velocity** slider is set to an equal value. You should not add more than one motor of the same type to the same component.
Linear Spring	Springs apply a force to a component. A spring with a higher spring constant will move a component faster than a spring with a lower spring constant. Also, a component with a smaller mass will move faster than a component with a larger mass if acted upon by springs of equal strength. Motion due to a spring stops when the spring reaches its free length. Motion due to motors supersedes motion due to springs. If you have a motor moving a component to the left and a spring pulling a component to the right, the component moves to the left.

Simulation Element	Description
Gravity	You can define only one gravity simulation element per assembly. All components move at the same speed under the effect of gravity regardless of their mass. Motion due to motors supersedes motion due to gravity. If you have a motor moving a component up and gravity pulling a component down, the component moves up without any downward pull.

Animation Controller

The Animation Controller is invoked by the **Replay Simulation** button on the Simulation toolbar.

Playback Options

There are several options for replaying the simulation:

- **Start**
- **Rewind**
- **Play**
- **Fast Forward**
- **End**
- **Pause**
- **Stop**
- **Save as AVI**
- **Normal**
- **Loop**
- **Reciprocate**
- **Slow Play**
- **Fast Play**

Progress Bar

FeatureManager Design Tree

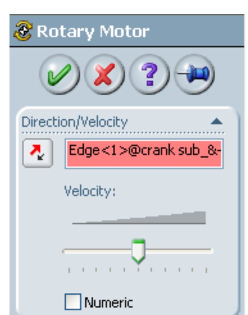
When you add simulation elements to an assembly, a `Simulation` feature is added to the FeatureManager. If you right-click on the `Simulation` feature, you can:

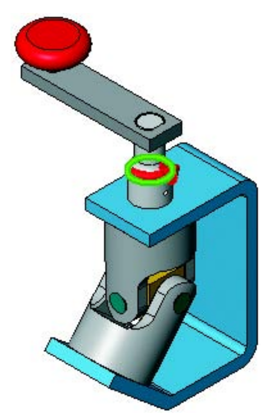
- Delete the `Simulation` feature, including all the simulation elements.
- Delete the replay of the simulation. This leaves the simulation elements intact.
- Reset the components to their positions prior to the simulation.

---

1 **Add a rotary motor.**

Click **Rotary Motor** and select the circular edge of the `crank-shaft` as the **Direction** of the motor.

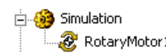




Tip

Clicking the **Numeric** option allows you to set a real value for the angular velocity in the currents units. In this example it would be degrees per second.

2 Simulation folder.



When the **Rotary Motor** is added, a new `Simulation` folder is added to hold it.

3 Calculate the simulation.

Click **Calculate Simulation** on the Simulation toolbar. Record approximately two complete revolutions of the `crank-assy`.

Note

When you record a simulation, the components actually move within their degrees of freedom according to the simulation elements. The degrees of freedom are determined by the mates on the components and collisions with other components.

4 Stop recording.

Click **Stop Record or Playback** on the Simulation toolbar.

5 Play the simulation.

Click **Replay Simulation** on the Simulation toolbar to access the Animation Controller. Use any of the controller options to speed up, slow down, loop or reciprocate the playback.

6 Save and close.

Another Example

The following section is another example of Physical Simulation where the motor is applied to one component and that motion affects several others.

---

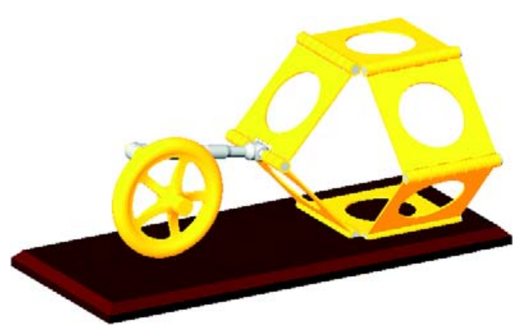
1 Open an assembly.

Open the assembly `machine.sldasm` located in the `Sarrus Mechanism` folder.

2 Hide Component.

Switch to the `Back Iso` view and hide the `Mount` component.

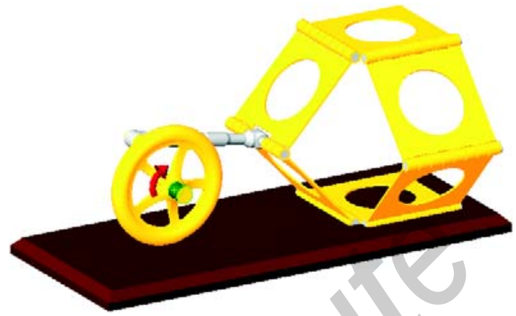
This will make it easier to add a rotary motor to the shaft of the `Wheel`.



3 Rotary Motor.
Click **Rotary Motor** on the Simulation toolbar.

Select the cylindrical face of the shaft of the `Wheel`.

Click **Reverse Direction** and then click **OK**.



4 Show Component.
Switch back to the `My Iso` view, and show the `Mount` component. Record, stop and play the simulation in the same fashion as the previous one.

Other Examples

You can use any of the assemblies in the `Physical Dynamics` folder to experiment with Physical Simulation.

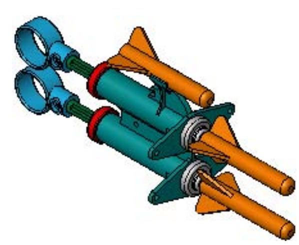
Exploded Assemblies

You can make **Exploded Views** of assemblies automatically or by exploding the assembly component by component. The assembly can then be toggled between normal and exploded view states. Once created, the **Exploded View** can be edited and also used within a drawing. **Exploded Views** are saved with the active configuration.

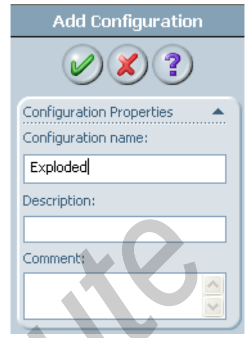
Setup for the Exploded View

Before adding the **Exploded View**, there are some setup steps that will make the exploded view easier to access. It is good practice to create a configuration for storing an **Exploded View** and also to add a mate the holds the assembly in a “starting position”.

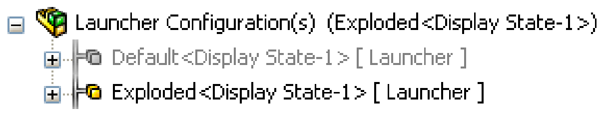
1 Open an assembly.
Open the assembly `Launcher.sldasm` located in the `Exploded View` folder.



2 Add a new configuration.
Switch to the ConfigurationManager, right-click and select **Add Configuration**.



Type the name `Exploded` and add the configuration.



The new configuration is the active one.

For more information on *Assembly Configurations*, see the *Advanced Assembly Modeling* manual.

Introducing: Exploded View

Exploded View is used to move one or more components along an arm of the **Move Manipulator** or triad. Each move direction and distance is stored as a step.

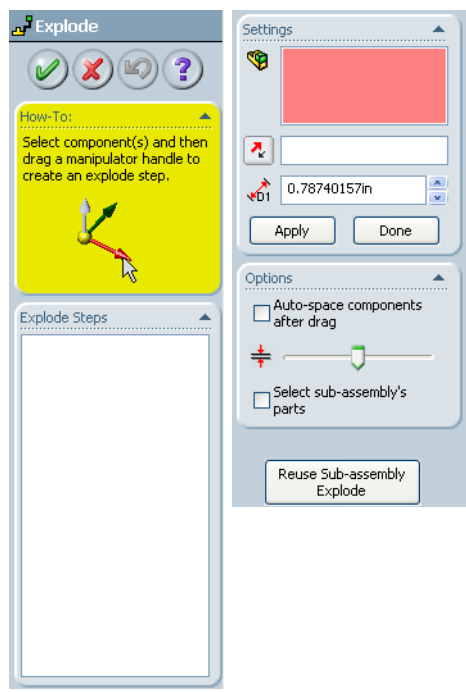
Where to Find It

- From the **Insert** menu, pick **Exploded View...**.
- Or, click **Exploded View** on the Assembly toolbar.

3 Click Insert, Exploded View.
The **Exploded View** dialog box appears. **Explode Steps** allows for individual movement of each component.

The **Settings** group box lists the components exploded in the current step along with direction and distance.

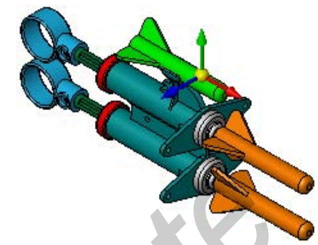
The **Options** group box includes the automation Auto-space and sub-assembly options.



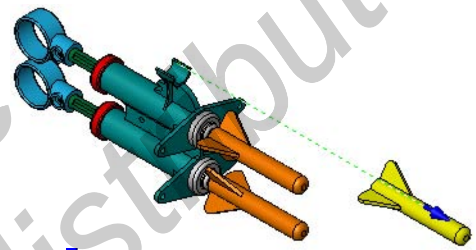
Exploding a Single Component

One or more components can be moved in one or more directions. Each movement (one or more components) set by a distance and direction is considered one step.

4 Select component.
Select the `Arrow<3>` component one the screen. A Move Manipulator appears at the center of the component bounding box. The Move Manipulator is aligned with the x leg along the length of the cylindrical face.



5 Drag explode.
Explode the component by dragging the red leg away from the assembly. The `Explode Step1` feature is added. The component is listed beneath it.

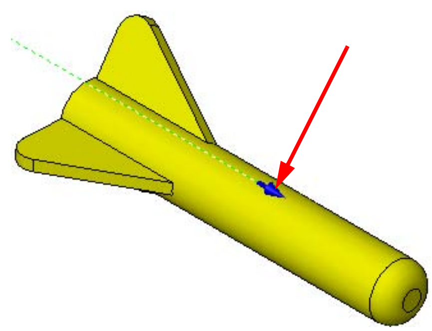


Click off the component to complete the step.

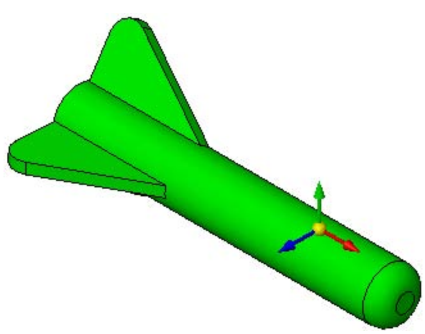
Tip Selecting the step by name in the dialog displays the components in yellow with the blue arrow.

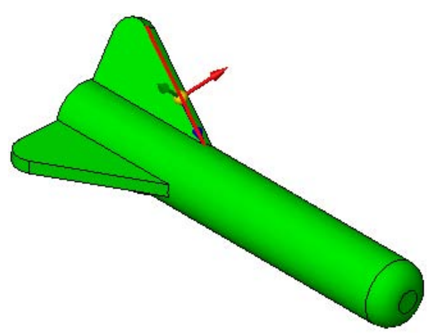
Move Manipulator and Drag Arrow

The Move Manipulator axes are used as vectors for the explode step. Once created, the step distance can be modified by dragging the blue arrow along the explode line.



If the Move Manipulator axes do not point in the desired directions, its orientation can be changed. Drag the manipulator origin and drop it on an edge, axis, face or plane to reorient it.

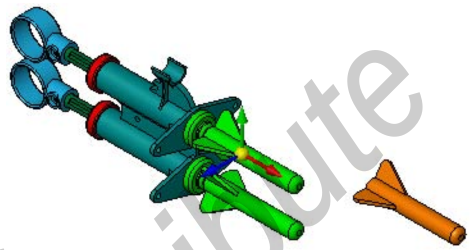




Multiple Component Explode

Multiple components can be exploded along the same path or multiple paths. For multiple component selections, the *last* component selected determines the orientation of the Move Manipulator.

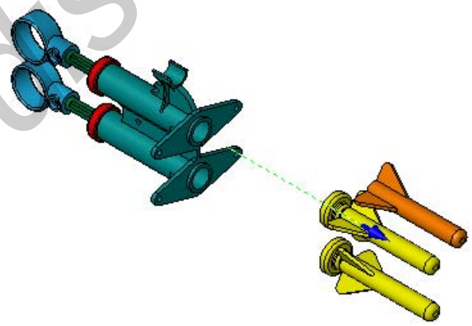
6 Selection.
Select the `Arrow<1>` component first followed by the remaining `Arrow` and both `Nozzle` components.



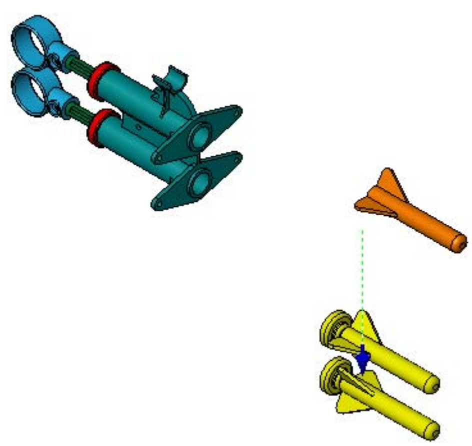
Tip

Making a multiple component selection can be made by clicking each one or using a drag-select window.

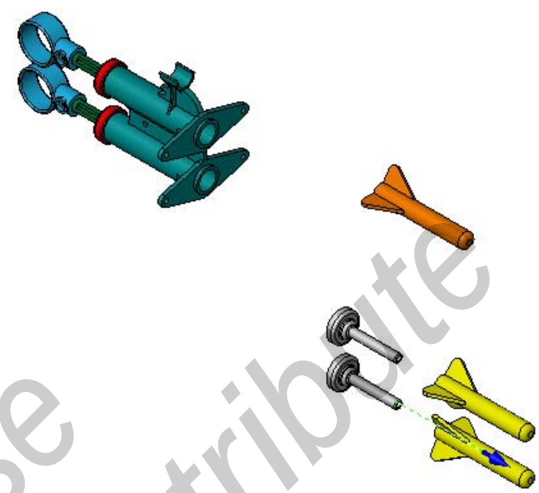
7 Paths.
Move the components along the red leg as shown.



Re-select the same components and add another step.



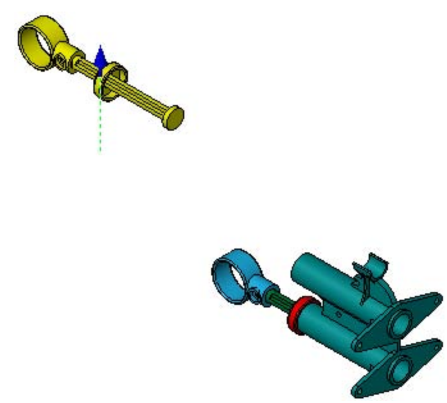
Add another step for just the `Arrow` components.



Sub-assembly Component Explode

Sub-assemblies can be treated in several ways. As single components, they can be moved as one. As individual components, each can be moved independently. If an exploded view of that sub-assembly already exists, it can be added to the current exploded view.

8 **Sub-assembly as component.**
Select the upper `SUB_trigger` component and move it with two steps as shown.

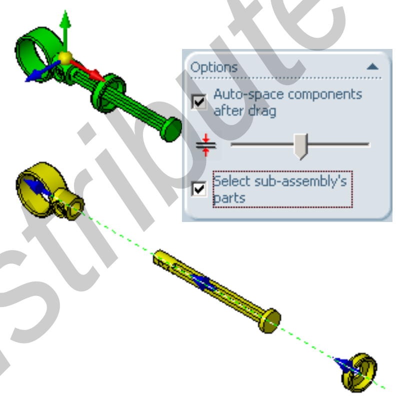


Auto-spacing

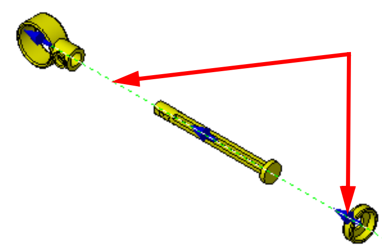
The **Auto-space components after drag** option is used to spread a series of components along a single axial step. The spacing can be set with a slider and changed after creation.

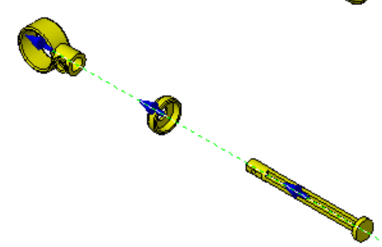
The **Select sub-assembly's parts** option treats each sub-assembly component as an individual component.

9 Auto-space.
Click **Select sub-assembly's parts** and **Auto-space components after drag**. Select the three components of the sub-assembly individually. Drag along the red leg and drop to space the components. A **Chain** step is added.



10 Adjust.
Drag the `End Cap` component using the blue arrow and drop it between the other two components. The spacing remains with a different order.



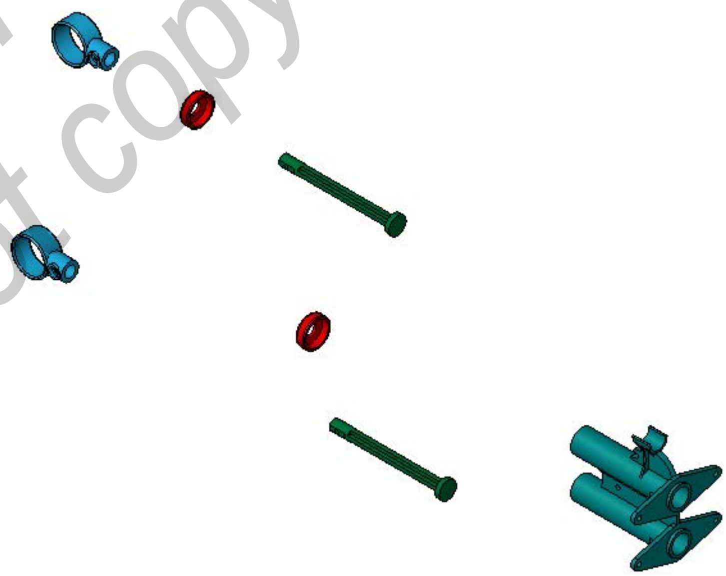


Reusing Explodes

Exploded views created within sub-assemblies can be imported and re-used.

11 Move sub-assembly.
Clear the **Select sub-assembly's parts** and **Auto-space components after drag** options. Drag the lower `SUB_trigger` sub-assembly as shown.

12 Reuse sub-assembly explode.
Select the sub-assembly and click **Reuse sub-assembly explode**.



Explode Line Sketch

Create lines as paths for the exploded view using **Explode Lines**. A type of 3D sketch called an **Explode Line Sketch** is used to create and display the lines. The **Explode Line Sketch** and **Jog Line** tools can be used to create and modify the lines.

Explode Lines

Explode Lines can be added to the explode line sketch to represent the explode path of the components.

Introducing: Explode Line Sketch

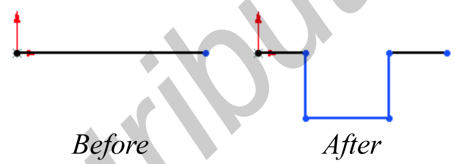
An **Explode Line Sketch** allows you to semi-automatically create explode lines. To do this, you select model geometry such as faces, edges, or vertices, and the system generates the explode lines.

Where to Find It

- On the **Insert** menu, click **Explode Line Sketch**.
- Or, click **Explode Line Sketch** on the Assembly toolbar.

Introducing: Jog Line

Jog Line is used to break an existing line and create a series of 90° lines. The jog lines are automatically constrained to be perpendicular and parallel to the original lines.



Where to Find It

- On the **Tools** menu, click **Sketch Tools, Jog Line**.
- Or, click **Jog Line** on the Explode Sketch toolbar.

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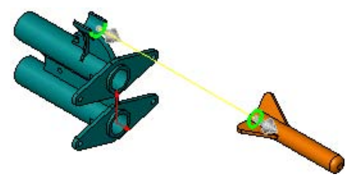
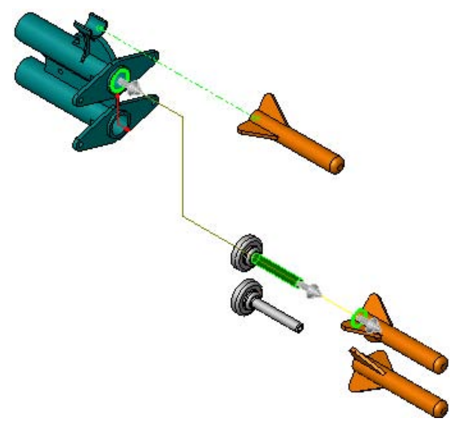
13 Route line.
Click **Explode Line Sketch** to start the 3D sketch. Select the arc and circle edges as shown to create a route line between them. Various combinations of the **Options** can be used to get different results.

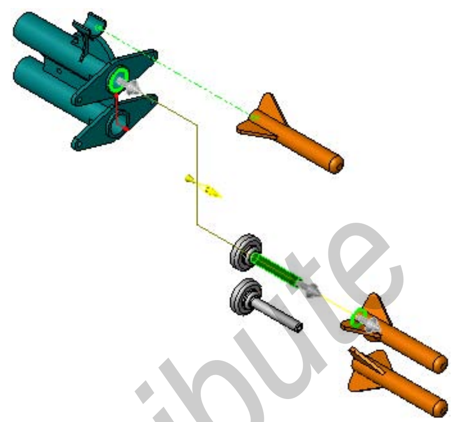


Click **OK**.

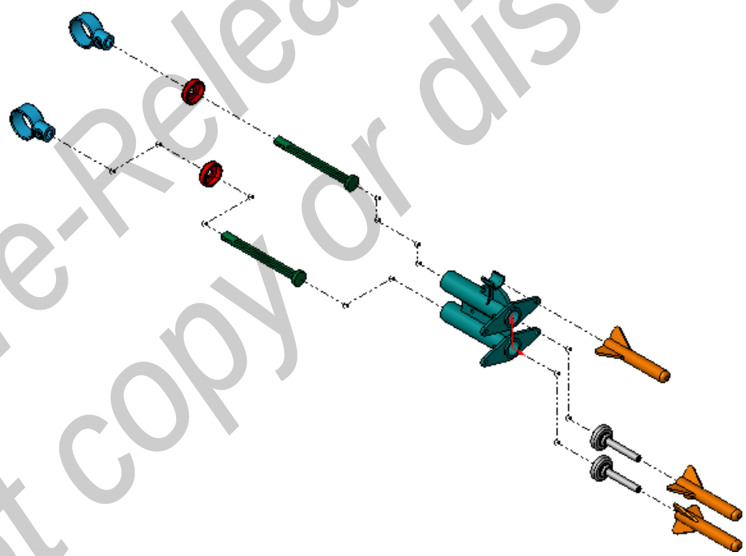
14 Explode through component.
Select (in order) the circular edge of the `Main Body<1>`, the cylindrical face of the `Nozzle<1>` and the circular edge of the `Arrow<1>`. A continuous series of explode lines is created.



15 Edit explode lines.
Move individual line segments by dragging the small arrows as shown. You can also click the small 3D arrows at the ends of the explode lines to reverse their direction. Right-click **OK**.



16 Additional lines.
Add sets of explode lines to connect all the remaining components.



Customizing Toolbars

The **Route Line** and **Jog Line** tools are located on the Explode Sketch toolbar by default. You may decide it is more convenient to add these two tools to an existing toolbar, such as the Sketch Tools or Assembly toolbar. You can do this by clicking **Tools, Customize...**, and selecting the **Commands** tab. Select **Explode Sketch** from the **Categories** list, and drag the buttons onto the desired toolbar.

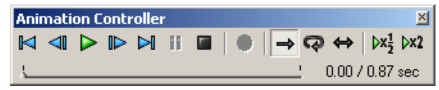
Animating Exploded Views

The Animation Controller can be used to animate the explode or collapse motion.

Where to Find It

- Right-click **Animate Collapse** from the `ExplodeView1` feature.
- If the exploded view is collapsed, right-click **Animate Explode** from the `ExplodeView1` feature.

17 Animation toolbar.
Right-click on `ExplView1` and choose **Animate Collapse**.



The dialog uses standard VCR-like controls including **Play**.

For more information on the other options, see *Animation Controller* on page 434.

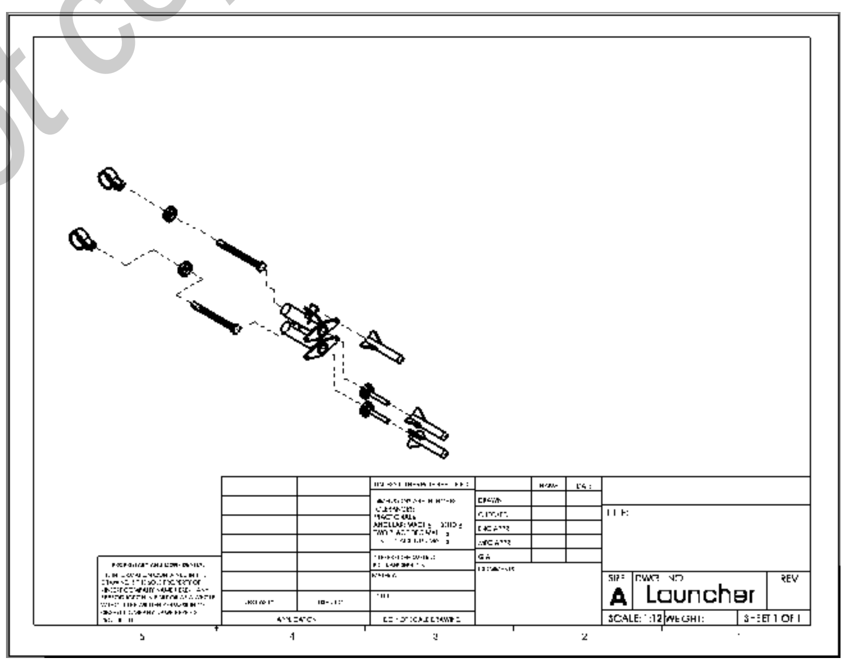
18 Save and close.
Collapse the assembly. **Save** and **Close** the assembly.

Assembly Drawings

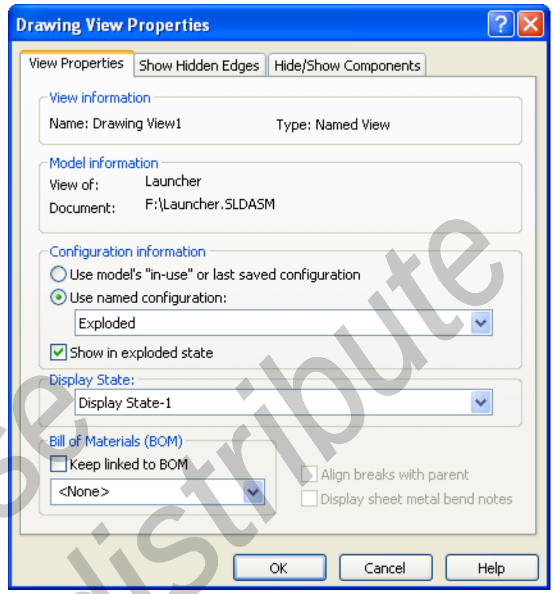
Assemblies have several unique requirements when it comes to making detail drawings of them. In addition to specialized views, assemblies require a Bill of Material and Balloons to fully document the assembly.

1 New drawing.
Use **Make Drawing from Part/Assembly** to create a new drawing using the `A-Scale1to2` template.

2 Named View.
Using the automatic **Model View** dialog, add a named view of the assembly `Launcher`. Set the orientation to `*Isometric` with **Scale 1:4** and place it on the drawing. This view will be used to display an exploded view.



3 View Properties.
Click **More Properties** to see that the `Exploded` configuration and **Show in exploded state** are selected.



Note

The **Show in exploded state** option will appear *only* if there is an existing exploded view in the selected configuration.

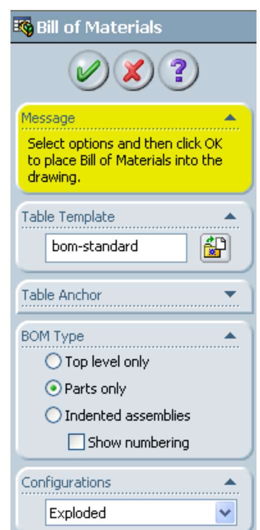
Bill of Materials

In an assembly drawing, a bill of materials report can be automatically created and inserted onto the drawing sheet.

Where to Find It

- Click **Bill of Materials** on the Table toolbar.
- Or from the **Insert** menu, select **Tables, Bill of Materials...**.

4 BOM Settings.
Click in the exploded view and click **Insert, Tables, Bill of Materials...**. Select `bom-standard` as the **Table Template**, **Parts only** as the **BOM Type** and `Exploded` as the **Configurations**.



Since both configurations contain the same components, either configuration will work.

5 Bill of Materials.

The BOM will appear on the screen where you clicked when you selected the view.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Plunger		2
2	Pull Ring		2
3	End Cap		2
4	Arrow		3
5	Nozzle		2
6	Main Body		1

You can resize the BOM columns by dragging them. The entire BOM can also be moved.

Adding Balloons

The item numbers assigned by the bill of materials can be added to the drawing using **Balloons**. These balloons will assign the proper item number as they are inserted onto edges, vertices or faces.

Introducing: Auto Balloons

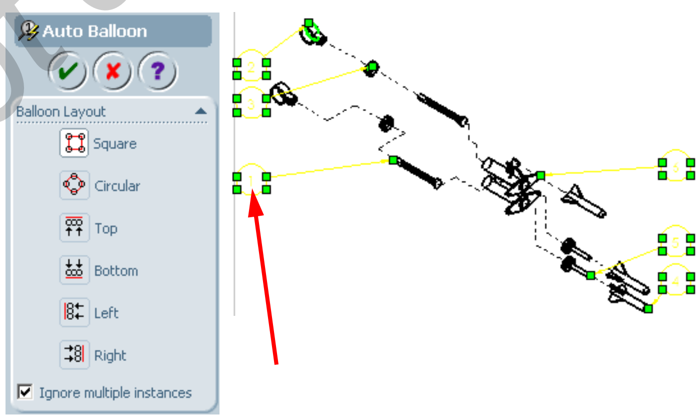
The **Auto Balloon** command is used to automatically label the components of an assembly drawing by item number and optionally quantity. There are several different shapes of balloons.

Where to Find It

- On the Annotations toolbar, click **Auto Balloon**.
- Or, on the **Insert** menu, click **Annotations, Auto Balloon...**.

6 Insert balloons.

Click on the **Auto Balloon** tool and select the **Square** layout. Balloons with the correct item numbers are added. When you drag the item **1** balloon closer to the view, they all move.



7 Save and close the drawing and any other open files.

In the Drawings Course...

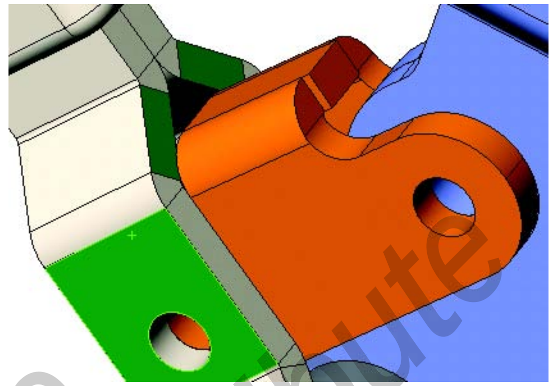
The three sections about drawings covered in this course are just an introduction to the art of detailing. In the course *SolidWorks Essentials: Drawings*, you will learn about all aspects of making detailed drawings of piece parts and assemblies.

Exercise 58: Using Collision Detection

Using the assembly provided, determine the range of motion of the clamp handle.

This lab reinforces the following skills:

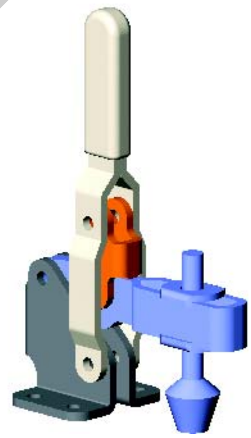
- **Collision Detection**.
- Dimensioning or using the **Measure** command.



Procedure

Open an existing assembly.

1 **Existing assembly.**
Open the existing assembly named `Collision` from the folder `Collision`.



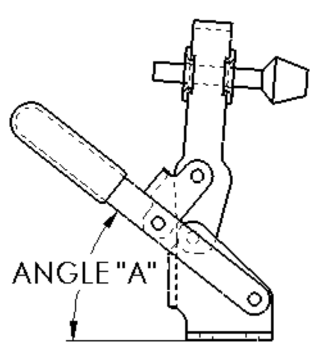
2 **Collision locations.**
The `link` stops the motion of the assembly in two places. Move the assembly to the point of collision and measure the angle formed using **Measure** or dimensions on a drawing view.

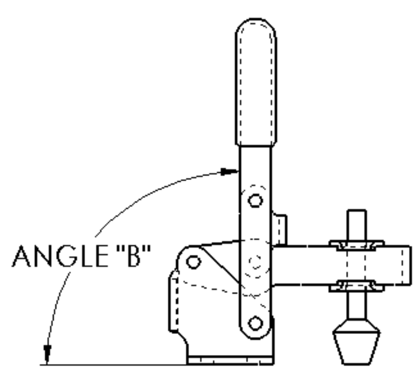
ANGLE “A”- As the `handle sub-assy` is pulled back, the `link` hits it.

ANGLE “B”- As the `handle sub-assy` is pushed forward, the `link` hits the `hold-down sub-assy`.

Measurements: (rounded)

Angle “A” = 38° **Angle “B” = 90°**





Exercise 59: Exploded Views

Using the existing assemblies, add exploded views and explode lines.

The files are found in the `Flashlight` folder.

Assembly: `Flashlight`

Tip

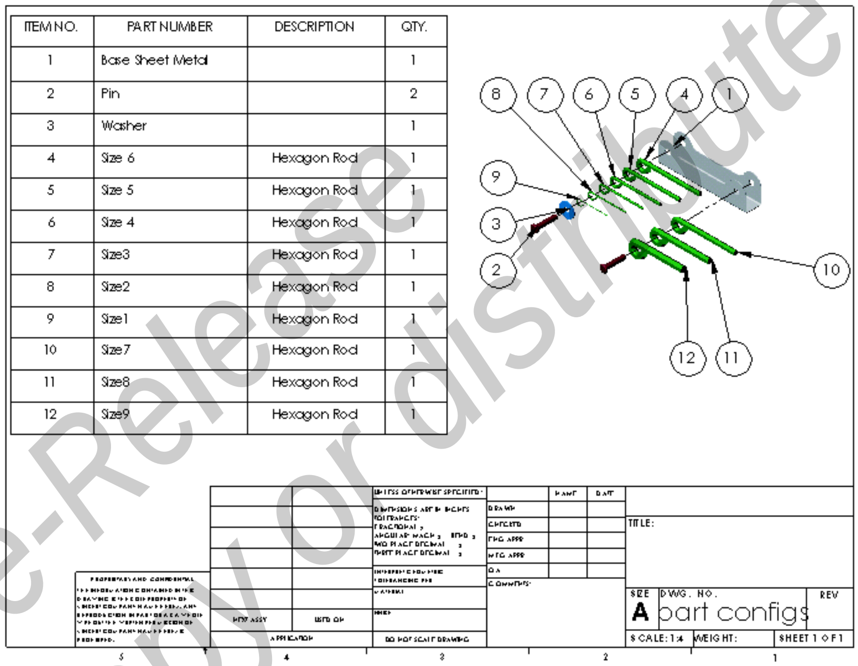
Many of the components are positioned on an angle and require the Triad to be dragged and dropped to set the proper explode direction. The triad can be dropped onto an edge or face.

Exercise 60: Exploded Views and Assembly Drawings

Using the existing assemblies, add exploded views and explode lines. Use the exploded views to generate drawings with balloons and a BOM. Use the `A-Scale1to2` template.

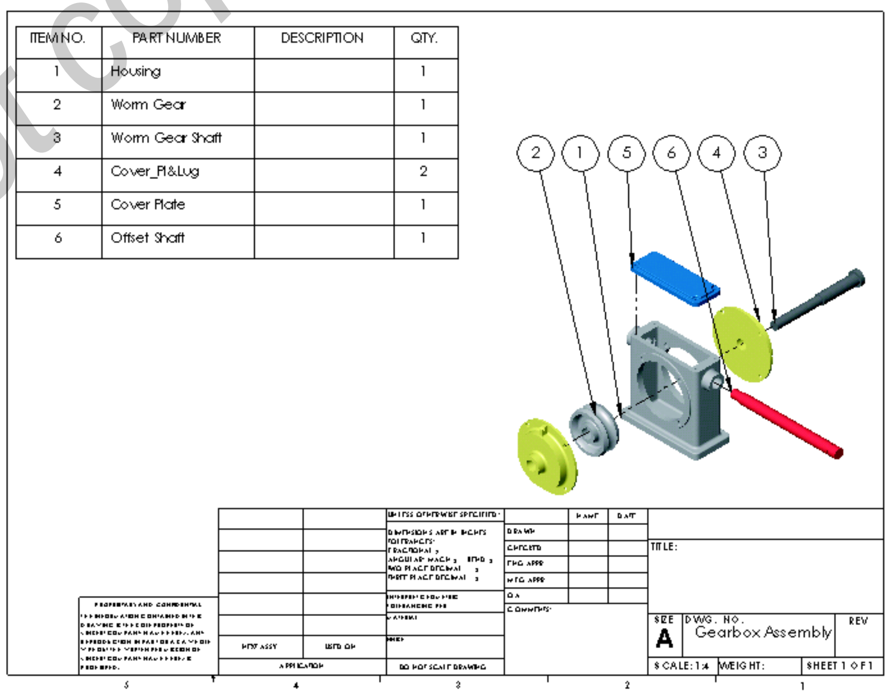
The files are found in the `Exploded Views` folder.

Assembly: `part configs`



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Base Sheet Metal		1
2	Pin		2
3	Washer		1
4	Size 6	Hexagon Rod	1
5	Size 5	Hexagon Rod	1
6	Size 4	Hexagon Rod	1
7	Size3	Hexagon Rod	1
8	Size2	Hexagon Rod	1
9	Size1	Hexagon Rod	1
10	Size7	Hexagon Rod	1
11	Size8	Hexagon Rod	1
12	Size9	Hexagon Rod	1

Assembly: `Gearbox Assembly`



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Housing		1
2	Worm Gear		1
3	Worm Gear Shaft		1
4	Cover_Pl&Lug		2
5	Cover Plate		1
6	Offset Shaft		1

Appendix

The material in this appendix supplements the material covered in the lessons. It was removed from the lessons to keep them of manageable length, and included here for your reference.

- **Tools, Options** settings used in this course.
- Creating a customized document template for parts.
- Organizing your document templates.

Options Settings

The **Tools, Options** dialog is the means by which default SolidWorks settings are changed. It contains settings that apply to individual documents and that are save with those documents, as well as settings that apply only to your system and your work environment.

The **Tools, Options** dialog contains two tabs that are labelled **System Options** and **Document Properties**.

Applying Changes

There are tabs within **Options** to make changes to the system or document properties. This allows you to control how the settings are applied. Your choices are:

- **System Options**
  Changes to the system options customize your work environment. They are not saved with specific document. Rather, any document opened on your system will reflect these settings. For example you might want your default spin box increment to be 0.25 inches. I might typically work on small parts and want a default spin box increment of only 0.0625 inches. System options let us each customize our work environment to our own needs.
- **Document Properties**
  Changes will affect only the currently open document. The system's default settings are not changed.

Changing the Default Options

To change the default **Options**, follow this procedure:

1. From the **Tools** menu, choose **Options**.
2. Select the tab for the settings you wish to change.
3. When finished, click **OK**.

Note You can only access document properties when a document is open.

Suggested Settings

For a complete listing of all the settings available through the **Tools, Options** dialog refer to the on-line help.

Important **System Options** that are used in this manual are:

- **General**
  Input dimension value: Enabled
  Maximize document on open: Enabled
- **Sketch**
  Display plane when shaded: Disabled
- **Default Templates**
  Always use these default document templates: Enabled

Document Templates

With a **Document Template** file (`*.prtdot`, `*.asmdot`, `*.drwdot`) you can save document properties for use in new documents. You can create a new template that contains just the settings that you want. When you want to create a new document, select the template you want and the document will inherit the template's settings.

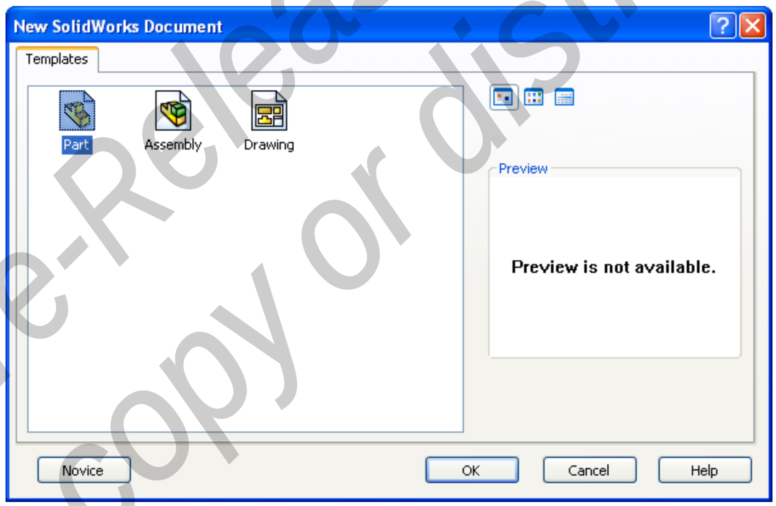
How to Create a Part Template

Creating a customized template is a simple procedure. You open a new document using the existing default template. Next you use the **Tools, Options** dialog to modify the document's settings. Then you save the document as a template file. You can set up folders to contain and organize your templates.

In this section we will create customized part template.

1 **Open a new part.**
Open a part using the default part template. The part will be used to form the template and will be discarded afterwards.

2 **Choose a template.**
Click **File, New** and the **Templates** tab of the dialog. Click the template `Part` and **OK**



Note

Do not use the **Novice** settings on the dialog when saving a document template. The resulting template will not be seen.

3 **Properties.**
Verify, and if needed, set the following **Document Properties**:

- **Detailing**
  Dimensioning Standard: ANSI
- **Detailing, Annotations Font**
  Dimension: Century Gothic; Height = 12 points
  Detail: Century Gothic; Height = 12 points
  Section: Century Gothic; Height = 12 points
  Note/Balloon: Century Gothic; Height = 12 points
- **Detailing, Dimensions**
  Precision, Primary Units, Value: 3
- **Detailing, Annotations Display**
  Always display text at same size: Enabled

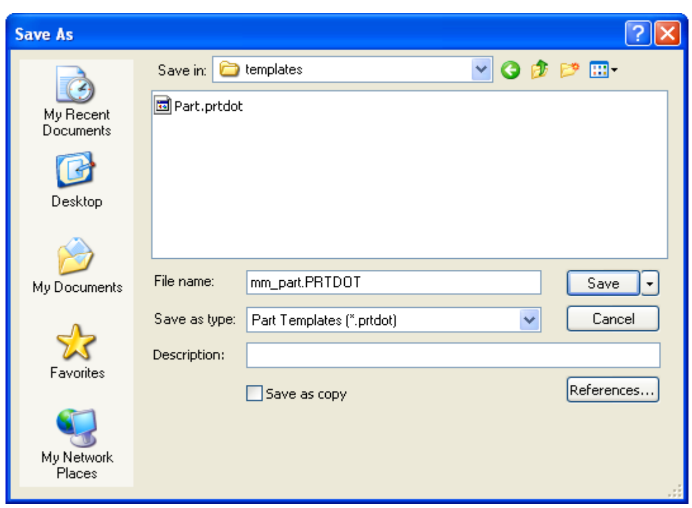
- **Grid/Snap**
  Display Grid - Disabled
  Snap to Points - Disabled
- **Units**
  Linear Units - Millimeters
- **Material Properties**
  Material properties are document-specific. Use **Edit Material** to set the proper material for the part. It is a good idea to create a part template for each commonly used material. This will save time and ensure accurate results when performing mass properties calculations and when making section views on drawings.
- **Reference Geometry**
  The default names for the three system reference planes are not controlled by **Tools, Options**. They are controlled by the document template. Since any feature can be renamed, the planes can be renamed as well. When the part is saved as a template, the plane names will be saved in the template file. Then, any new parts created using this template will automatically inherit the plane names. If you wish to, rename the reference planes. For example, you might prefer `XY`, `XZ`, and `YZ` instead of the default names.

4 **Save a template.**
Click **File, Save As**.

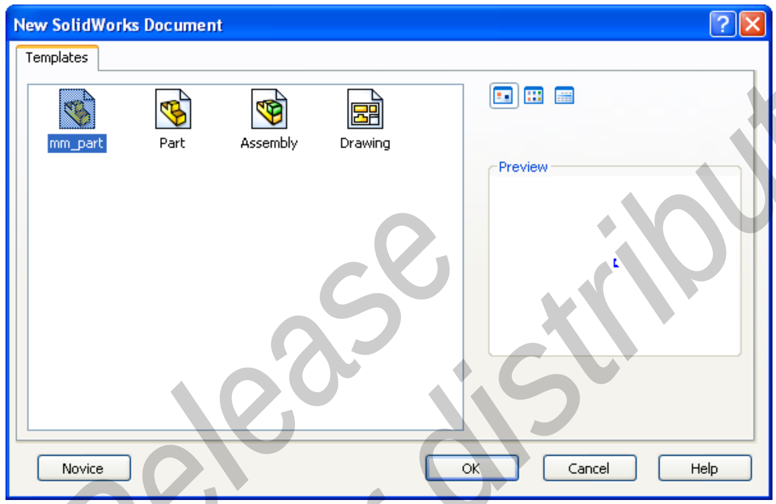
For **Save as type**, select **Part Templates**.

Name the template `mm_part` and navigate to the directory where you want to store your customized templates. In this example, we will simply save the template in the SolidWorks installation directory in the folder `Data\Templates`.

Click **Save**.



5 Use the template.
Close the current part without saving it. Open a new part using the template `mm_part` that appears in the dialog under the **Templates** tab. Check to see that the settings have been carried over.



Drawing Templates and Sheet Formats

Drawing templates and sheet formats have many more options than part or assembly templates. A complete treatment of creating and customizing drawing templates and sheet formats is covered in the course *SolidWorks Essentials: Drawings*.

Organizing Your Templates

As a general rule, it is not a good idea to store your customized templates in the SolidWorks installation directory. The reason for this is that when you install a new version of SolidWorks, the installation directory is overwritten. This would overwrite your customized templates.

A better strategy is to set up a separate directory for templates, just as you would for library features and standard parts libraries.

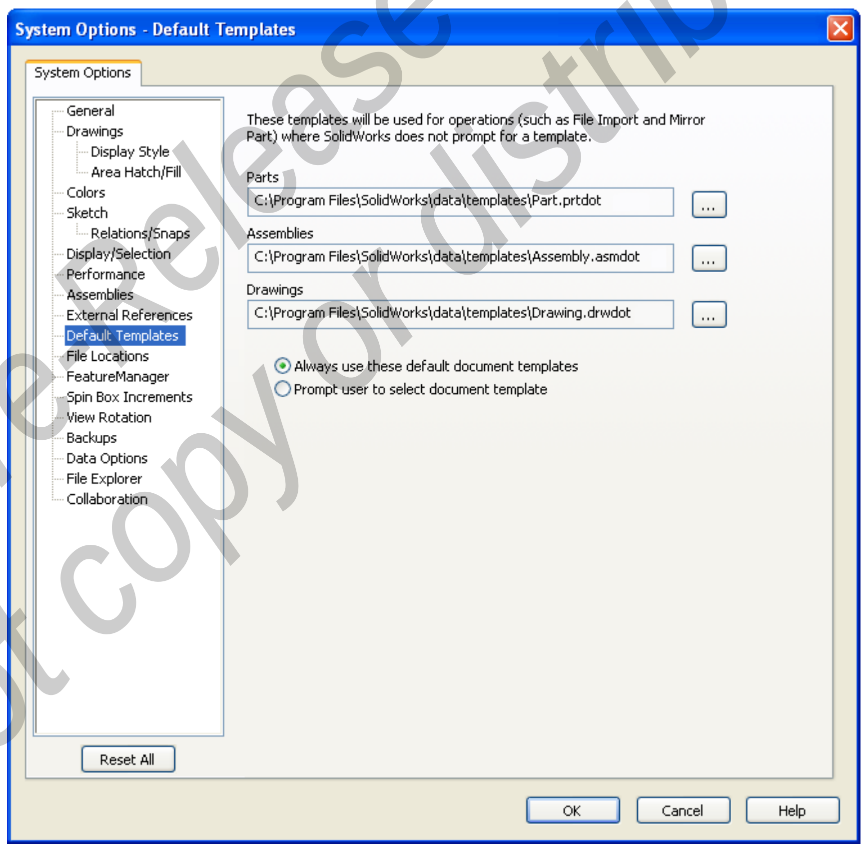
You can control where SolidWorks searches for the templates by means of **Tools, Options, System Options, File Locations**. The **Show folders for** box displays search paths for files of various types, including document templates. The folders are searched in the order they are listed. You can add new folders, delete existing folders, or move folders up or down to change the search order.

Default Templates

Certain operations in SolidWorks automatically create a new part, assembly, or drawing document. Some examples are:

- **Insert, Mirror Part**
- **Insert, Component, New Part**
- **Insert, Component, New Assembly**
- **Form New Sub-assembly Here**
- **File, Derive Component Part**

In these situations, you have the option of either specifying a template to use or having the system use a default template. This option is controlled by **Tools, Options, System Options, Default Templates**.



If you have selected **Prompt user to select document template**, the **New SolidWorks Document** dialog box will appear and you can choose the template you wish to use. If you have selected **Always use these default document templates**, the appropriate file will be automatically created using the default template. This section of the **Tools, Options** menu also enables you to define what template files the system should use by default.

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Question: What do you call an elite SolidWorks user?

Answer: A Certified SolidWorks Professional.

Get trained, get tested, and join our worldwide community of proven talent.

See reverse for more details.

SolidWorks Certified Professional

SolidWorks 2006

Certified SolidWorks Professional (CSWP) Planning Sheet

SolidWorks

Authorized Training, Testing & Support Center

Step	Task	Date Completed:
Step 1	**Attend Training:** SolidWorks Essentials - Parts and Assemblies	Date Completed:
Step 2	**Attend Training:** SolidWorks Essentials - Drawings	Date Completed:
Step 3	**Attend Training:** SolidWorks Advanced Part Modeling	Date Completed:
Step 4	**Attend Training:** SolidWorks Advanced Assembly Modeling	Date Completed:
Step 5	**Attend Training:** SolidWorks Sheet Metal and Weldments	Date Completed:
Step 6	Allow between 4 to 6 months "on the job" with SolidWorks	Date Completed:
Step 7	**Download CSWP Sample Exam** www.solidworks.com	Date Completed:
Step 8	**Take CSWP Exam**	Date Completed:

CSWPs receive each of the following:

Official Certificate

Custom Portfolio and Pen

Business Card Logos

SolidWorks Certified Professional

Exclusive Partner Discounts

SolidWorks World: Discounts, Special Events and Gear

SolidWorks World 2006
International User Conference and Exposition

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