SolidWorks® 2011

SolidWorks Essentials

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Introduction

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.

The SolidWorks software 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 The SolidWorks software. 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.
- Completed the online tutorials that are integrated in the SolidWorks software. You can access the online tutorials by clicking **Help, Online Tutorial**.

Course Length

The recommended minimum length of this course is 4 days.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. 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 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 Training Files

A complete set of the various files used throughout this course can be downloaded from the SolidWorks website, **www.solidworks.com**. Click on the link for **Support**, then **Training**, then **Training Files**, then **SolidWorks Training Files**. Select the link for the desired file set. There may be more than one version of each file set available.

Direct URL:

`www.solidworks.com/trainingfilessolidworks`

The files are supplied in signed, self-extracting executable packages.

The files 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.

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.

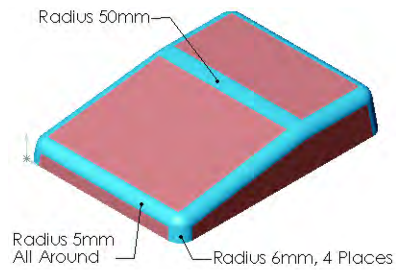
Windows® XP

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

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.



Graphics and Graphics Cards

The SolidWorks software sets a new standard with best-in-class graphics. The combination of a highly reflective material and the realism of **RealView Graphics** is an effective tool for evaluating the quality of advanced part models and surfaces.

RealView Graphics is hardware (graphics card) support of advanced shading in real time. For example, if you rotate a part, it retains its rendered appearance throughout the rotation.

Color Schemes

Out of the box, the SolidWorks software provides several predefined color schemes that control, among other things, the colors used for highlighted items, selected items, sketch relation symbols, and shaded previews of features.

We have not used the same color scheme for every case study and exercise because some colors are more visible and clear than others when used with different colored parts.

In addition, we have changed the viewport background to plain white so that the illustrations reproduce better on white paper.

As a result, because the color settings on your computer may be different than the ones used by the authors of this book, the images you see on your screen may not exactly match those in the book.

Lesson 1
SolidWorks Basics and the User Interface

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.

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 in the previous paragraph 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.

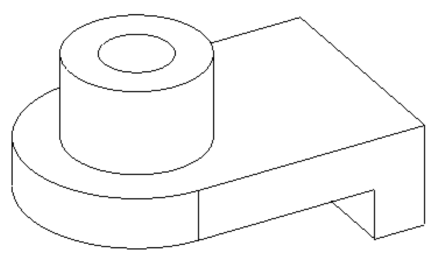
  When you create a model using the SolidWorks software, you work with intelligent, easy to understand geometric features such as bosses, cuts, holes, ribs, fillets, chamfers, and drafts. As the features are created they are applied directly to the work piece.

  Features can be classified as either sketched or applied.

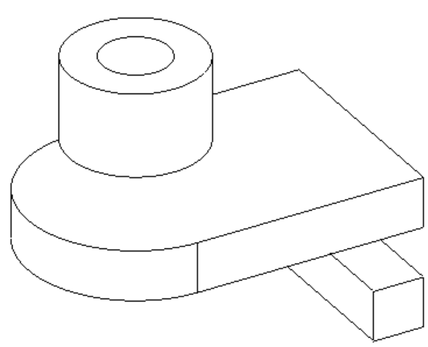
  - Sketched Features: Based upon a 2-D sketch. Generally that sketch is transformed into a solid by extrusion, rotation, sweeping or lofting.
  - Applied Features: Created directly on the solid model. Fillets and chamfers are examples of this type of feature.

  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.

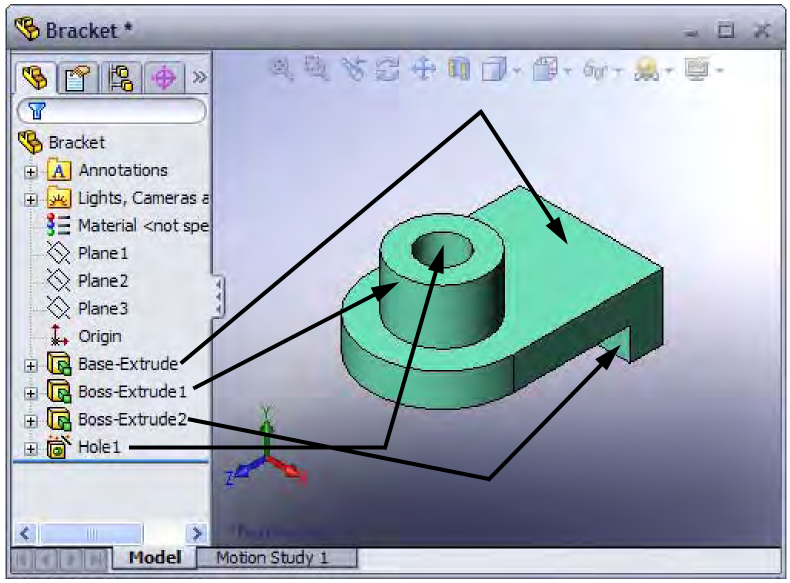
  To illustrate the concept of feature-based modeling, consider the part shown at the right:



  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 enables 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

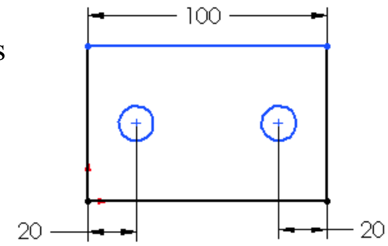
In order to use a parametric modeler like SolidWorks efficiently, you must consider the design intent before modeling. Design intent is your plan as to how the model should behave when it is changed. The way in which the model is created governs how it will be changed. Several factors contribute to how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

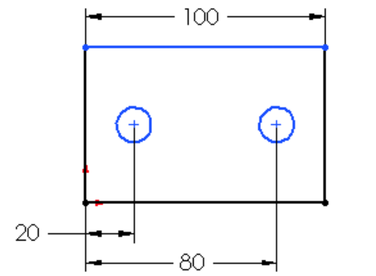
Examples of Design Intent

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

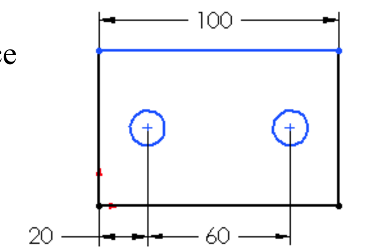
A sketch dimensioned like this will keep the holes **20mm** from each end regardless of how the overall plate width, **100mm**, is changed.



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.

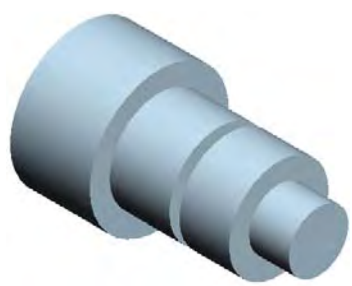


Dimensioning from the edge and from center to center will maintain the distance between the hole centers and allow it to be changed that way.



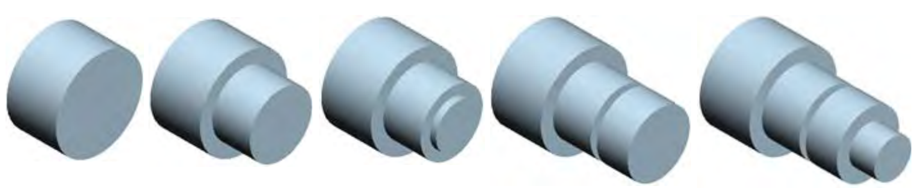
How Features Affect Design Intent

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

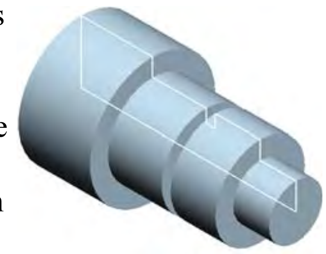
The layer cake approach builds the part one piece at a time, adding each layer, or feature, onto the previous one, like this:



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

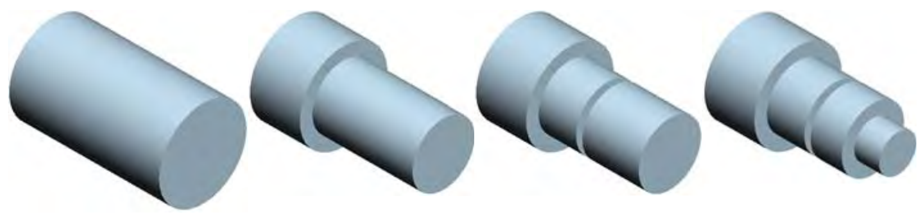
The “Potter’s Wheel” Approach

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.



File References

SolidWorks creates files that are compound documents that contain elements from other files. File references are created by linking files rather than duplicating information in multiple files.

Referenced files do not have to be stored with the document that references them. In most practical applications, the referenced documents are stored in multiple locations on the computer or network. SolidWorks provides several tools to determine the references that exist and their location.

Object Linking and Embedding (OLE)

In the Windows environment, information sharing between files can be handled either by linking or embedding the information.

The main differences between linked objects and embedded objects are where the data is stored and how you update the data after you place it in the destination file.

Linked Objects

When an object is linked, information is updated only if the source file is modified. Linked data is stored in the source file. The destination file stores only the location of the source file (an external reference), and it displays a representation of the linked data.

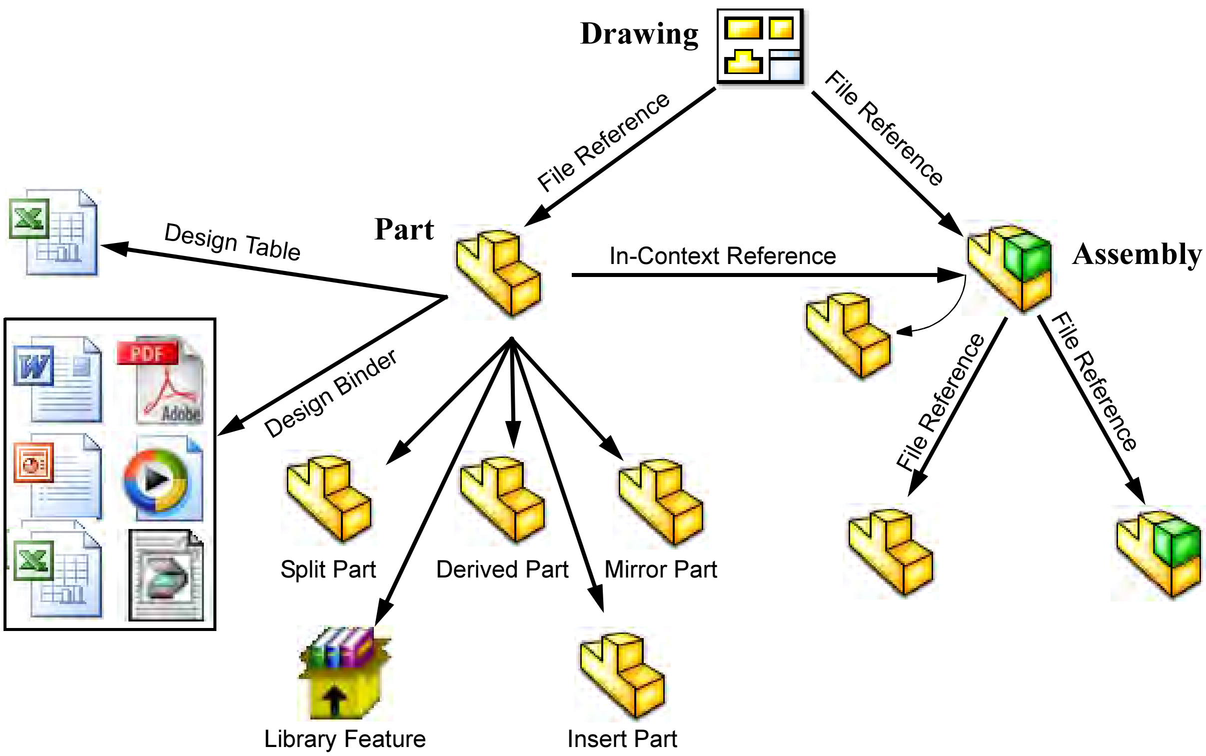
Linking is also useful when you want to include information that is maintained independently, such as data collected by a different department.

Embedded Objects

When you embed an object, information in the destination file doesn't change if you modify the source file. Embedded objects become part of the destination file and, once inserted, are no longer part of the source file.

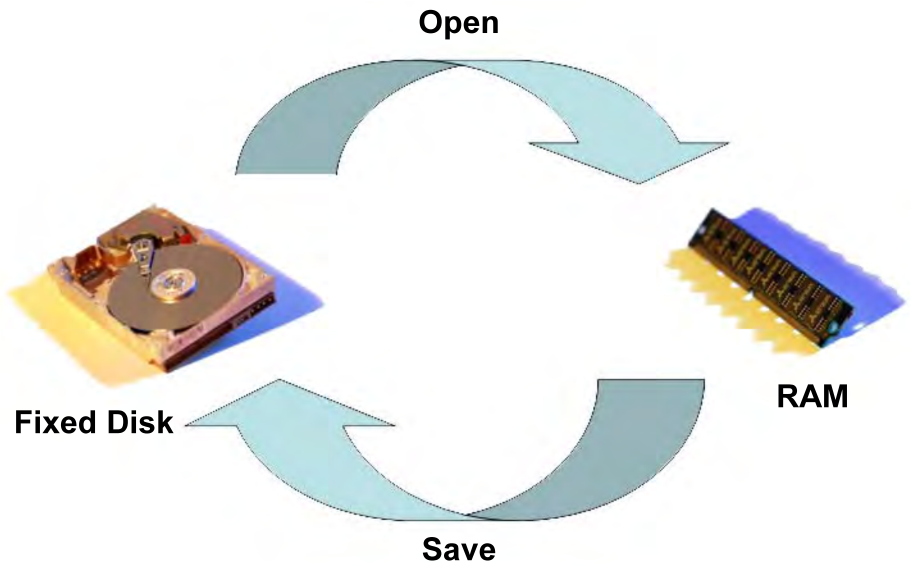
File Reference Example

The many different types of external references created by SolidWorks are shown in the following graphic. Some of the references can be linked or embedded.



Opening Files

SolidWorks is a RAM-resident CAD system. Whenever a file is opened, it is copied from its storage location to the computer's Random Access Memory or RAM. All changes to the file are made to the copy in RAM and only written back to the original files during a **Save** operation.



Computer Memory

To better understand where files are stored and which copy of the file we are working on, it is important to differentiate between the two main types of computer memory.

Random Access Memory

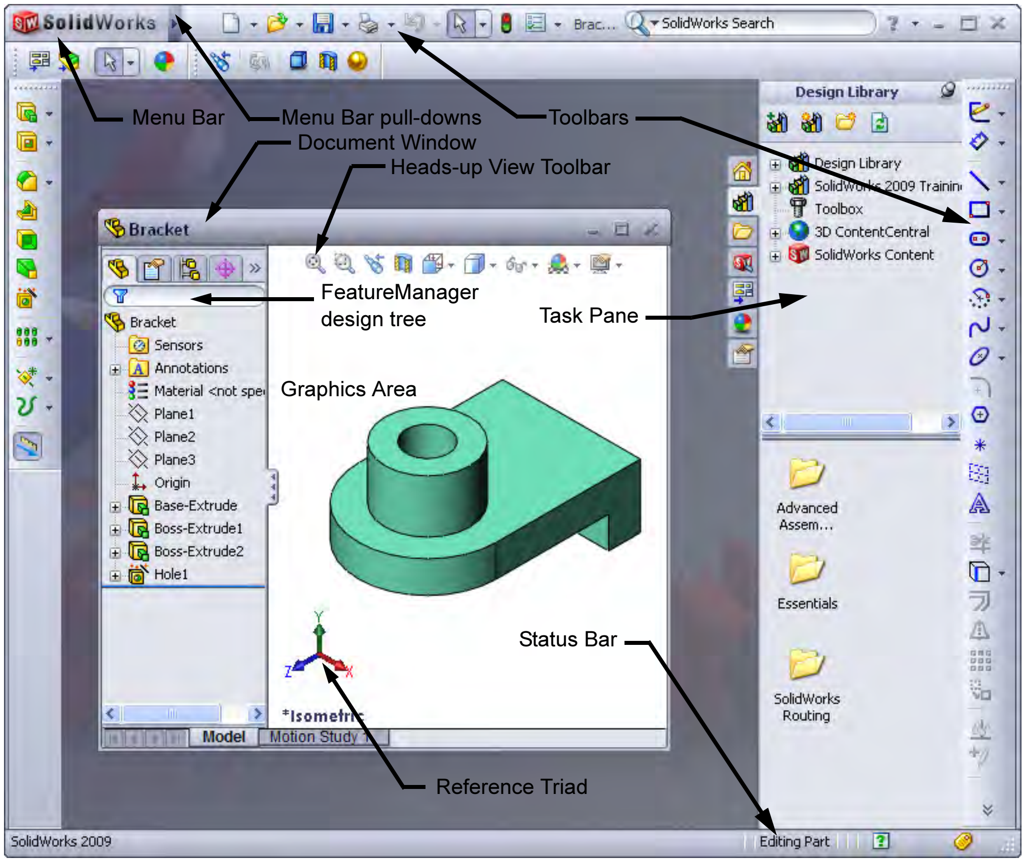
Random Access Memory (RAM) is the volatile memory of the computer. This memory only stores information when the computer is operating. When the computer is turned off, any information in RAM is lost.

Fixed Memory

Fixed memory is all the non-volatile memory. This includes the computer hard drive, floppy disks, zip disks and CDs. Fixed memory holds its information even when the computer is not running.

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.



Unselectable Icons

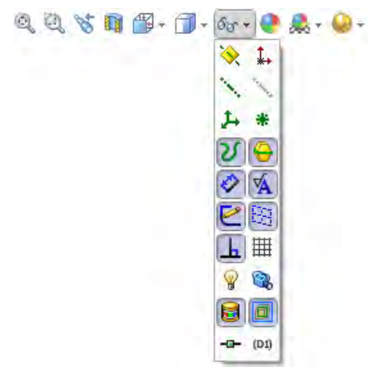
At times you will notice commands, icons, and menu options that are grayed out and unselectable. This is because you may not be working in the proper environment to access those options. For example, if you are working in a sketch (**Edit Sketch** mode), you have full access to all the sketch tools. However, you cannot select the icons such as fillet or chamfer on the Features toolbar. Likewise, when you are not working in a sketch, 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.

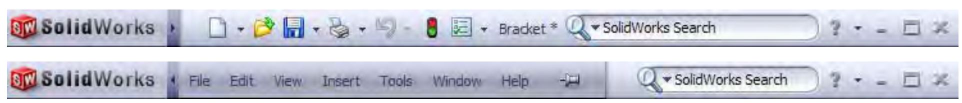
Heads-up View Toolbar

The **Heads-up View** toolbar is a transparent toolbar that contains many common view manipulation commands. Many of the icons (such as the **Hide/ Show Items** icon shown) are **Flyout Tool** buttons that contain other options. These flyouts contain a small down arrow  to access the other commands.



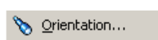
Pull-down Menus

The Pull-down menus provide access to many of the commands that the SolidWorks software offers. Float over the right facing arrow to access the menus. Click the pushpin to keep the menus open.

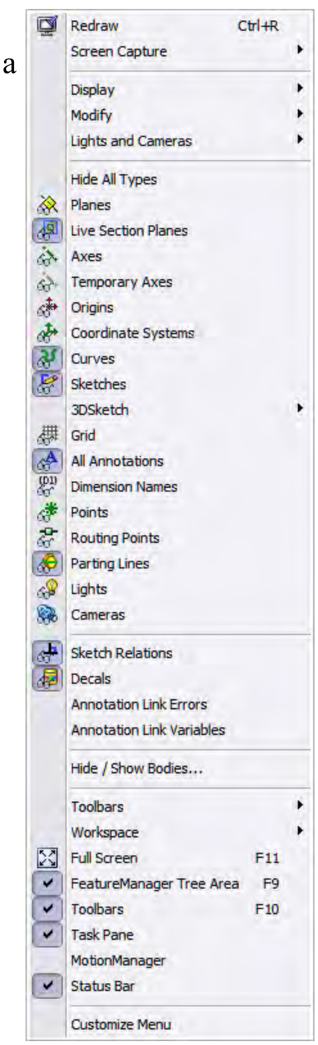


When a menu item has a right-pointing arrow like this: 

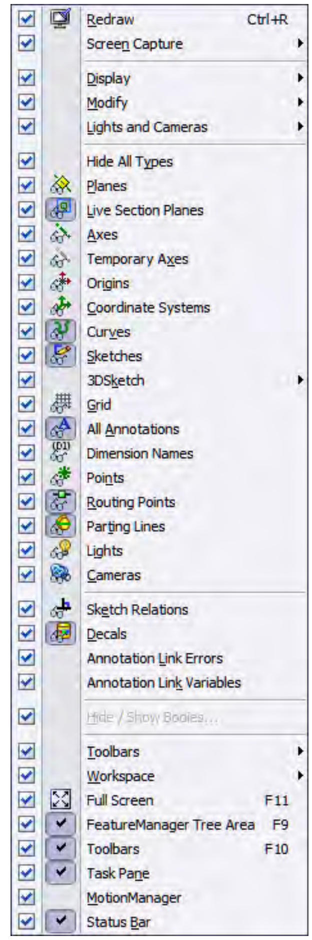
, it means that there is a sub-menu associated with that choice.

When a menu item is followed by ellipses like this: 

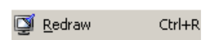
, it means that the option opens a dialog box with additional choices or information.



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: 

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 toolbars provide icon 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 icons 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 at right. This toolbar contains commonly used functions.

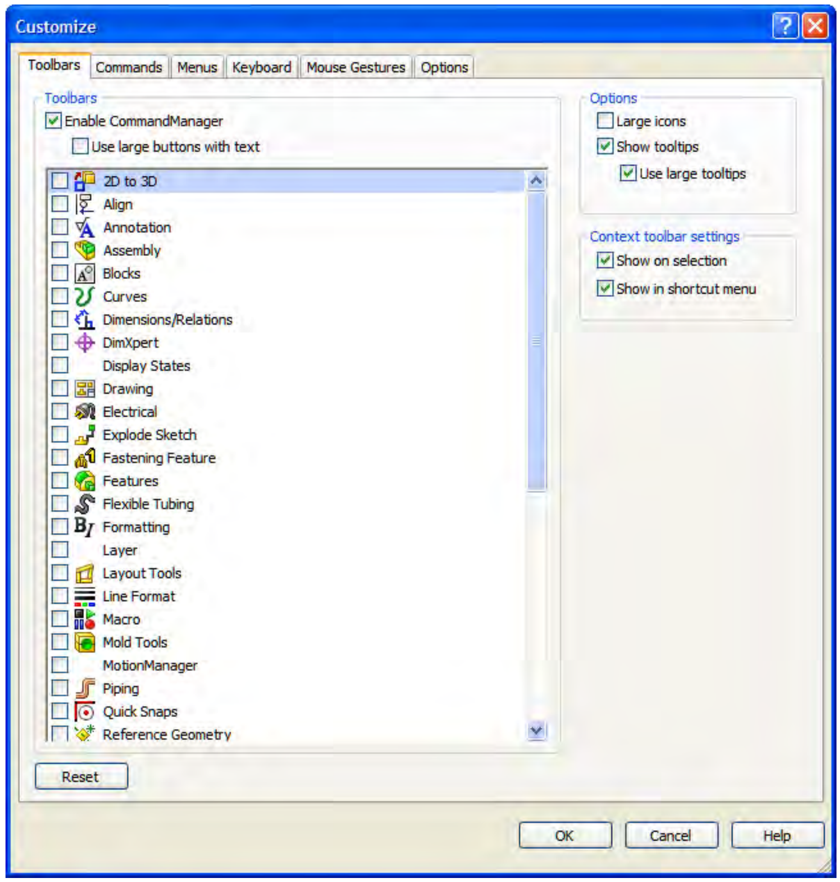
Flyouts

Many toolbars can be replaced with a single icon that contains all the toolbar icons. These flyout icons (**Rectangle** shown here)  have both an icon image and a pull-down to access other similar icons. The last used icon appears in the flyout.

Making Toolbars Visible

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

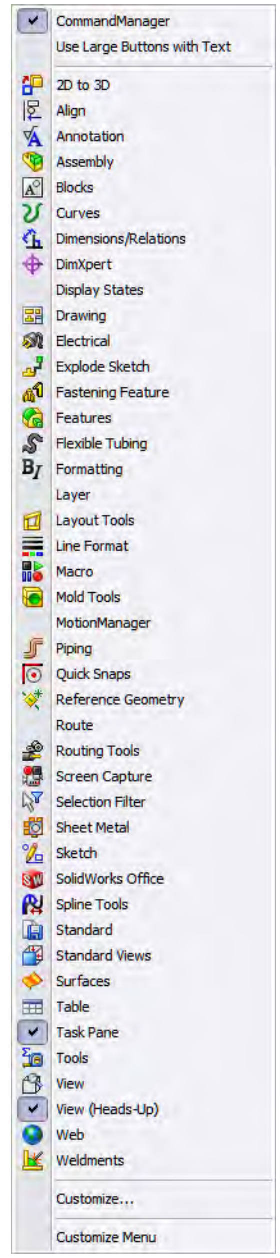
- **Click Tools, Customize.**
  On the **Toolbars** page, click the check boxes to select each toolbar you want to display. Clear the check boxes of the toolbars you want to hide.



Note

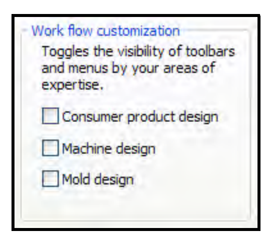
In order to access **Tools, Customize**, you must have a document open. Also the **Commands** tab can be used to add or remove icons from toolbars.

- **Right-click in the toolbar area of the SolidWorks window.**
  Pressed icons indicate which toolbars are currently visible. Click the toolbars you want to see.
- **Click View, Toolbars.**
  This displays the same list of toolbars.



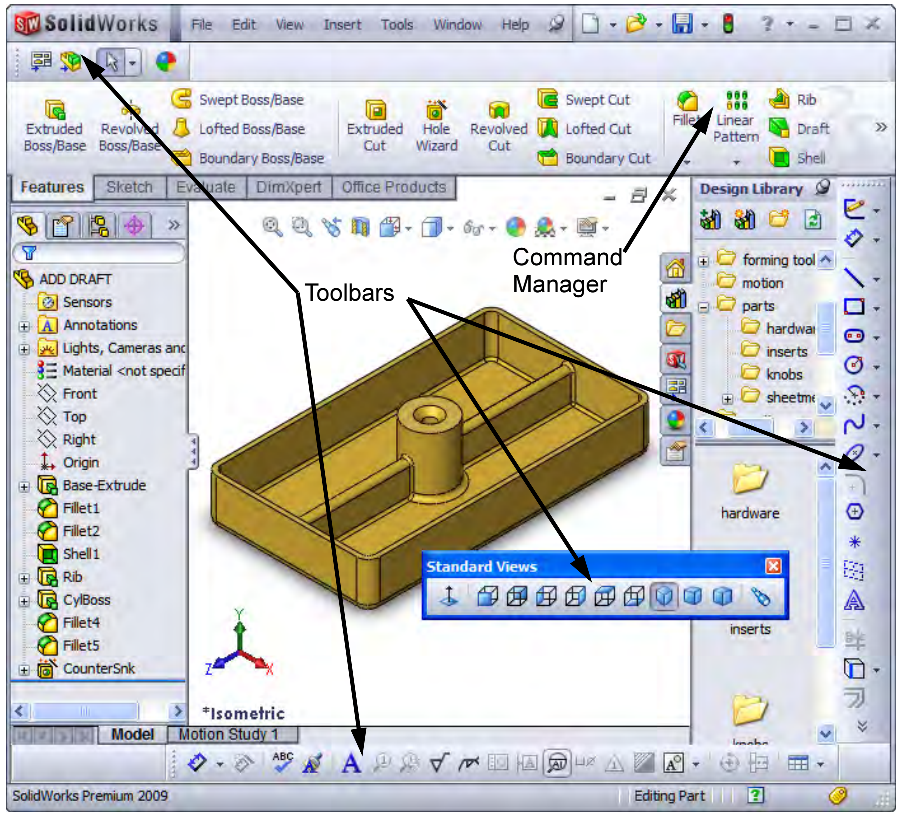
Work flow Customization

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



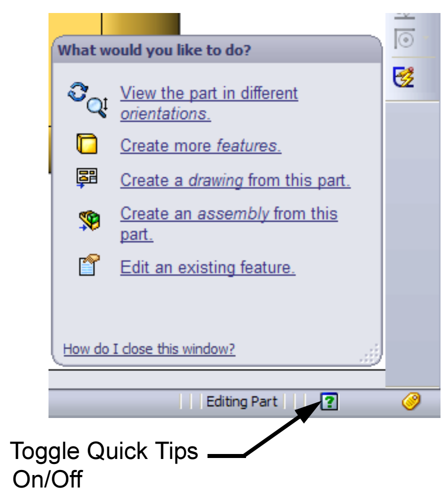
Arranging the Toolbars

The toolbars 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, including the CommandManager, is shown below.



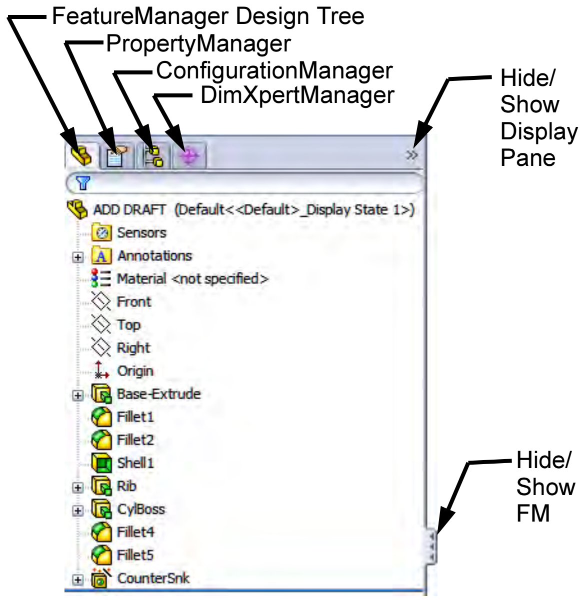
Quick Tips

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.



FeatureManager Design Tree

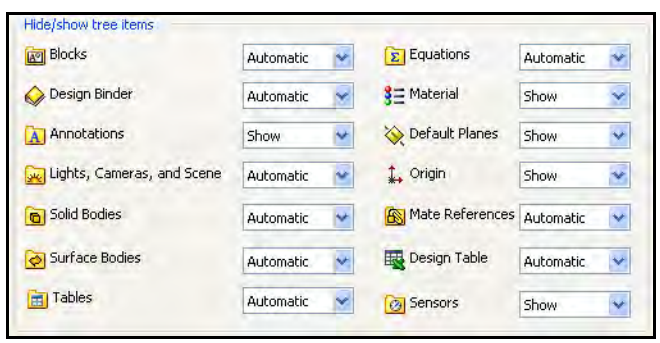
The FeatureManager design tree is a unique part of the SolidWorks software that visually displays all the features in a part or assembly. As features are created they are added to the FeatureManager design tree. As a result, the FeatureManager design tree represents the chronological sequence of modeling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



Show and Hide FeatureManager Items

Many FeatureManager items (icons and folders) are hidden by default. In the image above, only two folders (Sensors and Annotations) are shown.

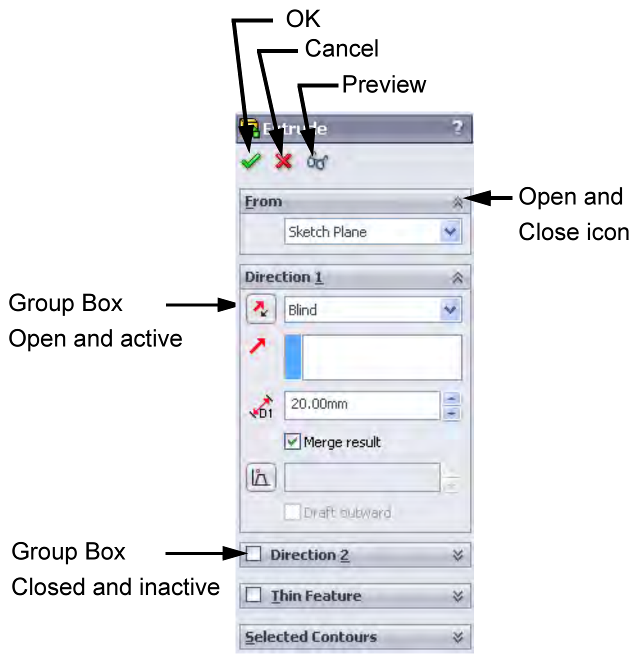
Click **Tools**, **Options**, **System Options**, and **FeatureManager** to control their visibility using one of the three settings explained below.



- **Automatic** - Hides the item when it is empty.
- **Hide** - Hide the item at all times.
- **Show** - Show the item at all times.

PropertyManager

Many SolidWorks commands are executed through the PropertyManager. The PropertyManager occupies the same screen position as the FeatureManager design tree and replaces it when it is in use.



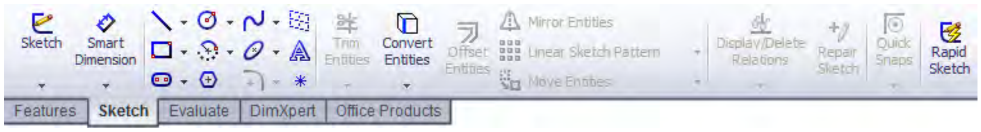
The top row of buttons contains the standard **OK**, **Cancel** and **Preview** buttons.

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.

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 several tabs to access commands related to **Features**, **Sketches** and so on.

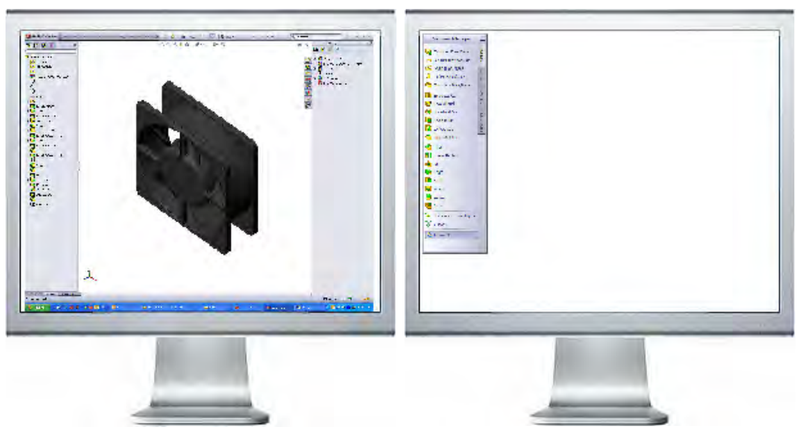
It can be displayed with or without text on the buttons.



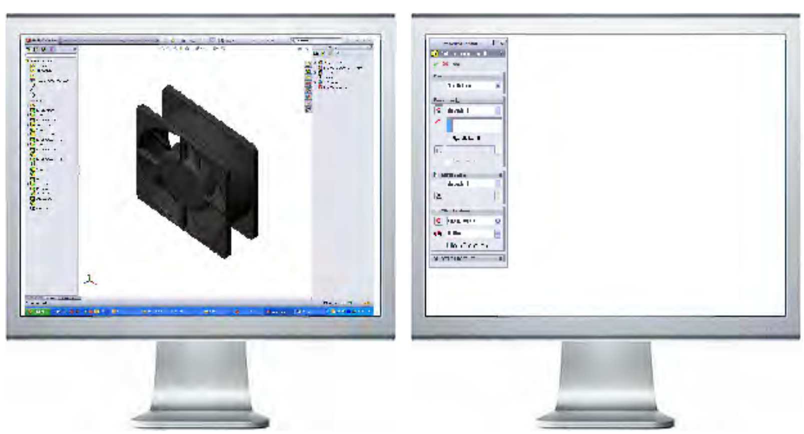
Moving the PropertyManager and CommandManager

The CommandManager or PropertyManager can be dragged and docked on the top, side or outside of the SolidWorks window. Outside the SolidWorks window can be a different monitor.

CommandManager

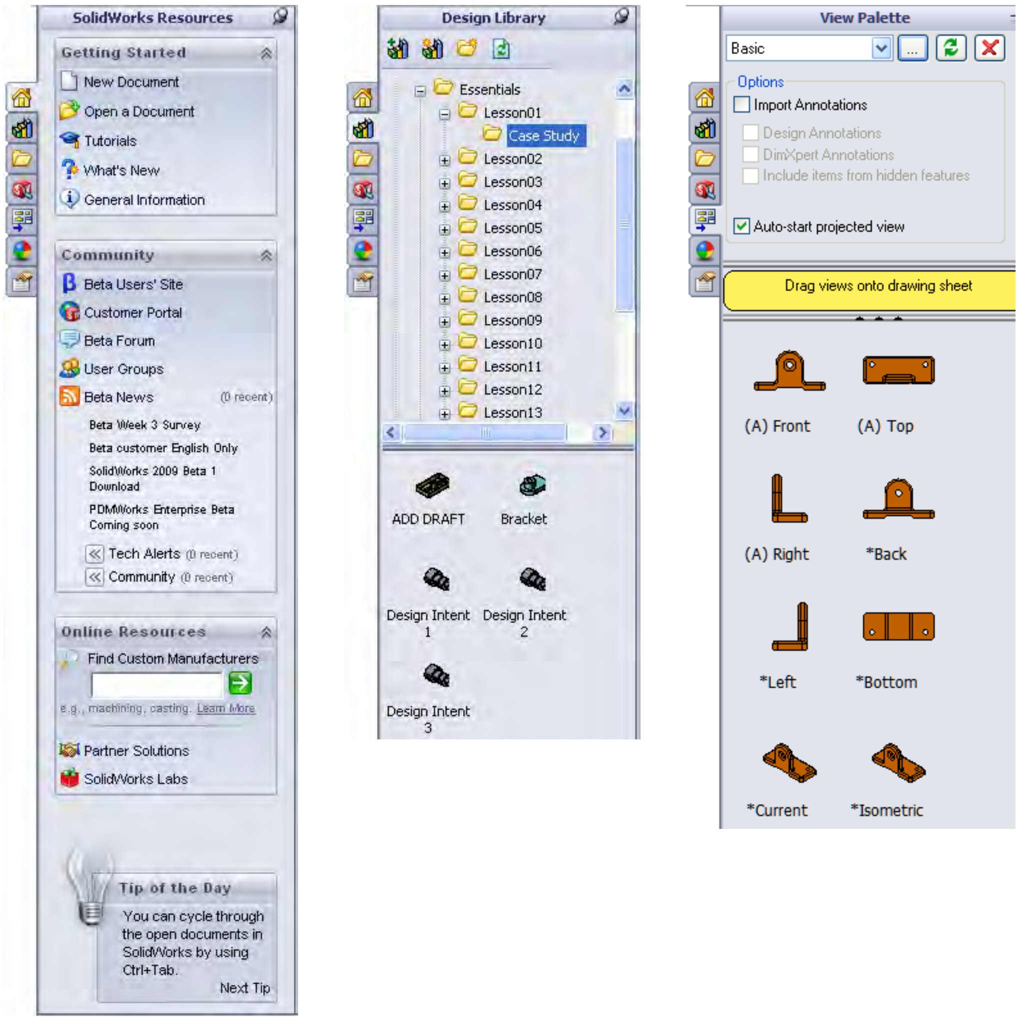


PropertyManager



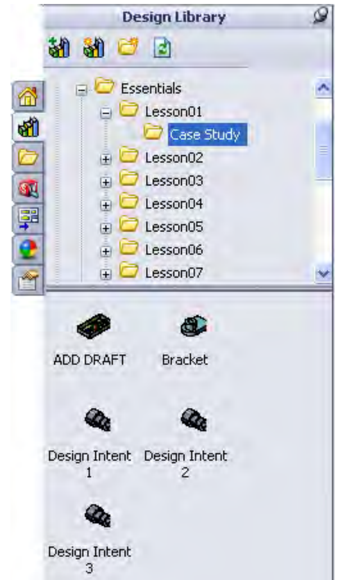
Task Pane

The **Task Pane** window is used to house the **SolidWorks Resources**, **Design Library**, **File Explorer**, **Search**, **View Palette**, **Appearances/Scenes** and **Custom Properties** 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 folder used for the class files. It should be found under the SolidWorks Training Files folder.
- Click **OK**.

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

SolidWorks Search

The **SolidWorks Search** option can be used to find files by searching for any part of the name. The Windows Desktop Search engine must be installed to use this feature. Search using this procedure.

- Type a name or partial name into the **SolidWorks Search** box and click the icon.
- The **Search** tab of the **Task Pane** is used to display the results.
- Open a file by clicking on the image and clicking **Open File in SolidWorks** .
- Click on the pathname text to open a search of that folder.

Mouse Buttons

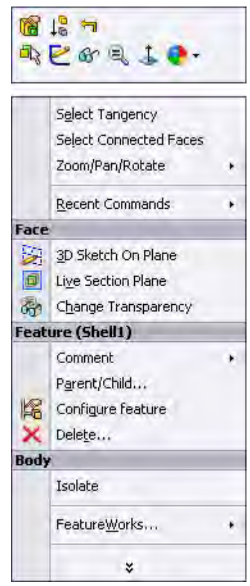
The left, right and middle mouse buttons have distinct meanings in SolidWorks.

- **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.

Note

At the top of the context sensitive shortcut menu is the **Context Toolbar**. It contains some of the most commonly used commands in icon form.

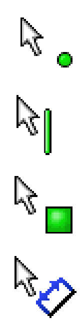
Below it is the pull-down menu. It contains other commands that are available in the context of the selection.



- **Middle**
  Dynamically rotates, pans or zooms a part or assembly. Pans a drawing.

System Feedback

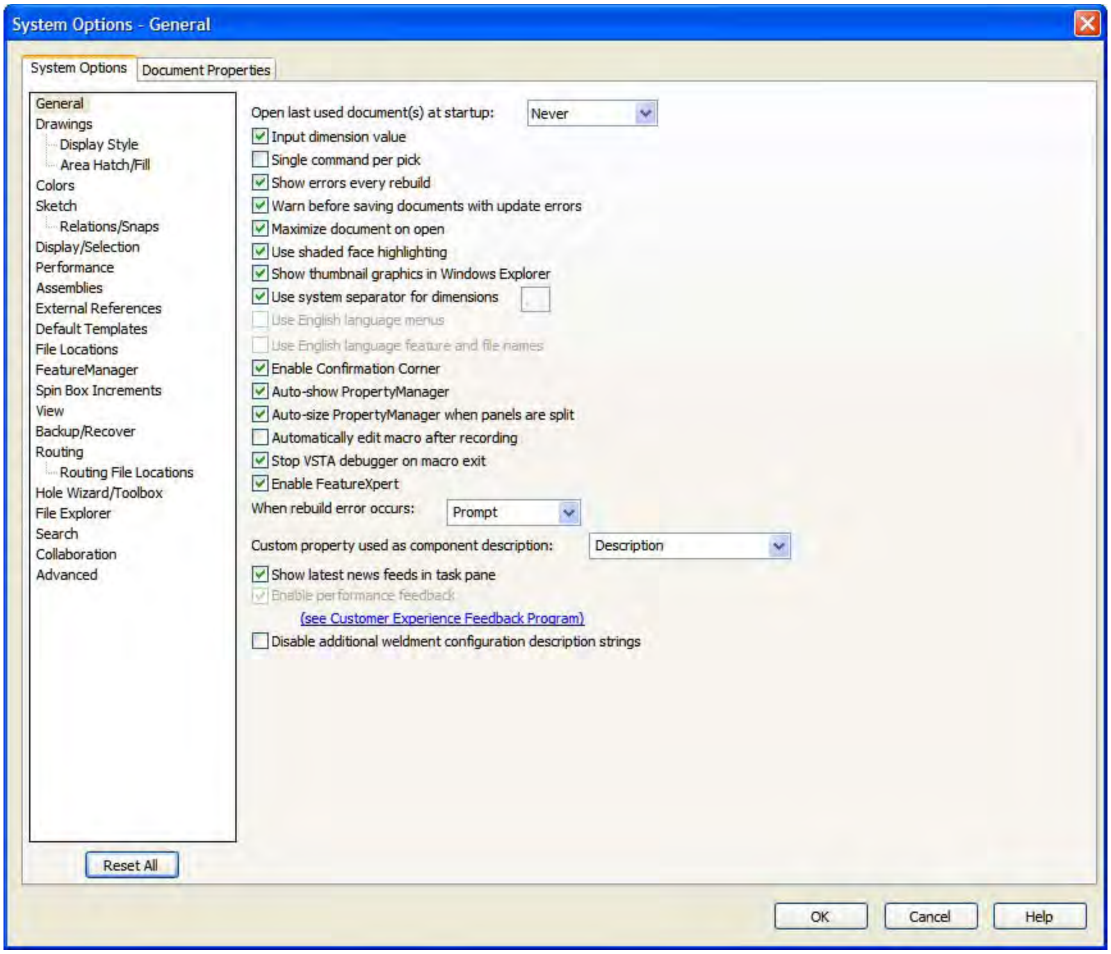
Feedback is provided by a symbol attached to the cursor arrow indicating what you are selecting or what the system is expecting you to select. As the cursor floats across the model, feedback will come in the form of symbols, riding next to the cursor. The illustration at the right shows some of the symbols: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



Vertex
Edge
Face
Dimension

Options

Located on the **Tools** menu, the **Options** dialog box enables 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**
  These settings are applied to the individual document. For example, units, drafting standards, and material properties (density) are all document settings. 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 482 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 482 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.

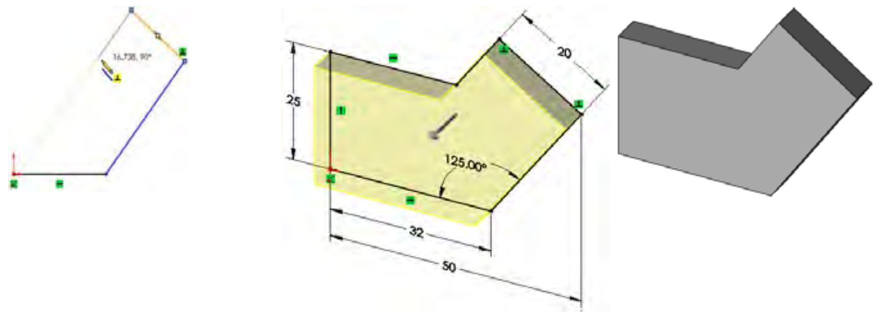
Lesson 2
Introduction to Sketching

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.
- Extrude the sketch into a solid.

2D Sketching

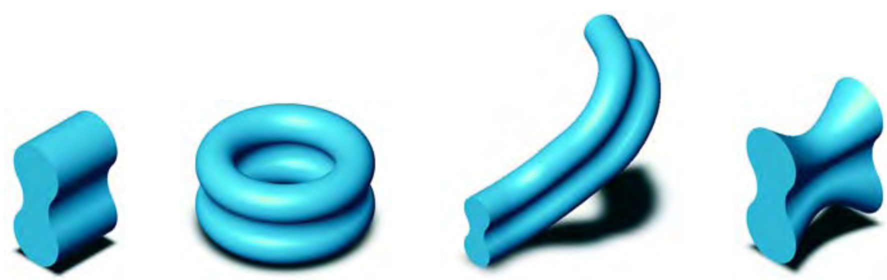
This lesson introduces 2D sketching, the basis of modeling in SolidWorks.



Sketches are used for all sketched features in SolidWorks including:

- Extrusions
- Revolves
- Sweeps
- Lofts

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



Extrude **Revolve** **Sweep** **Loft**

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 extending 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.

Introducing: New Part

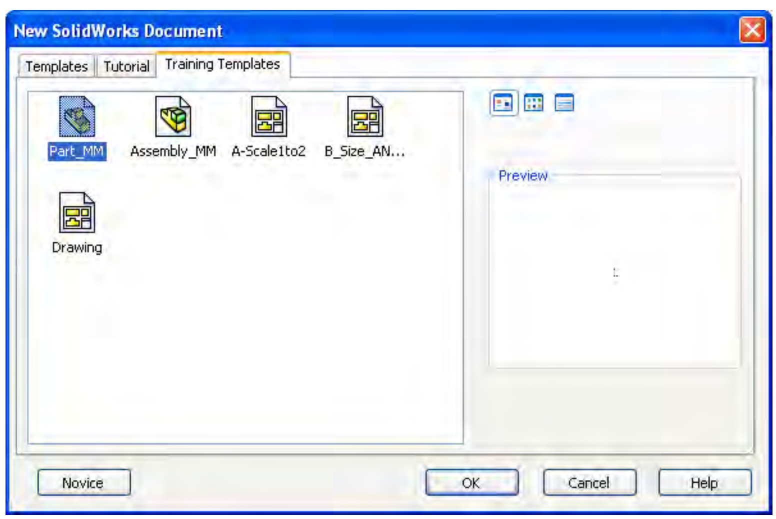
The **New** tool creates a new SolidWorks document from a selection of part, assembly or drawing templates. There are several training templates in addition to the default ones.

Where to Find It

- From the **File** menu, select **New**.
- Or, on the Standard toolbar, click **New**.

1 New part.

Click **New**, or click **File, New**. Click the `Part_MM` 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 millimeters as the units. You can create and save any number of different templates, all with different settings.

Saving Files

Saving files writes the file information in RAM to a location on a fixed disk. SolidWorks provides three options for saving files. Each has a different effect on file references.

Save

Copy the file in RAM to the fixed disk, leaving the copy in RAM open. If this file is being referenced by any open SolidWorks files, there are no changes to the reference.

Where to Find It

- Click **File, Save...**.
- Click **Save** on the Standard toolbar.

Save As

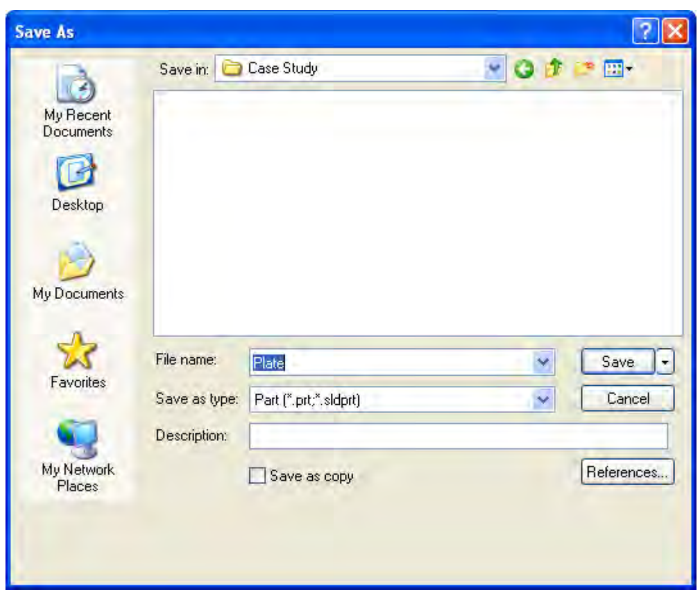
Copy the file in RAM to the fixed disk under a new name or file type, replacing the file in RAM with the new file. The old file in RAM is closed *without* saving. If this file is being referenced by any *open* SolidWorks files, you should update the references to this new file.

Save As Copy

Copy the file in RAM to the fixed disk under a new name or file type, leaving the copy in RAM open. If this file is being referenced by any open SolidWorks files, you *should not* update the references to this new file.

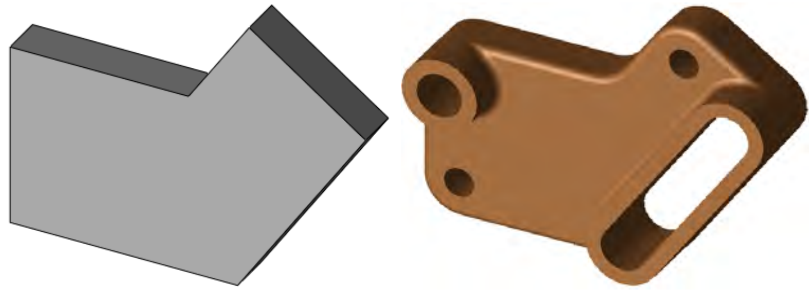
2 Filing a part.

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-D 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 Plane`, `Top Plane`, and `Right Plane`.

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.

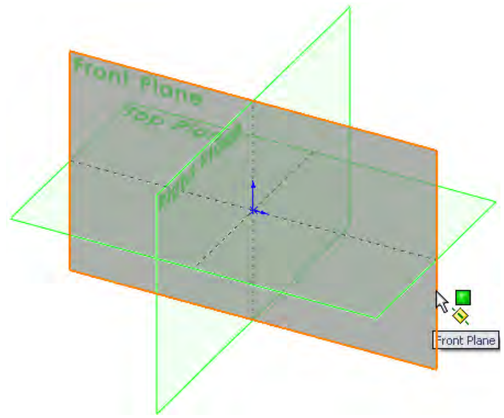
You must select a plane or a planar face of the model after clicking **Insert, Sketch**. The cursor appears indicating that you should select a face or plane.

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, click or right-click and choose **Insert Sketch** from shortcut menu.

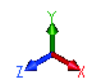
3 Open a new sketch.
Open the sketch by either clicking or choosing **Sketch** from the **Insert** menu. This will show all three default planes for selection in a Trimetric orientation. A Trimetric orientation is a pictorial view that is oriented so the three mutually perpendicular planes appear unequally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



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

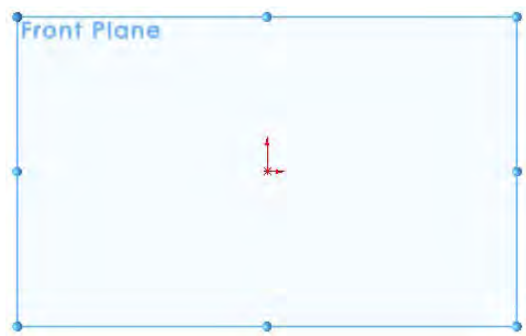
Note

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 sketch origin. It is displayed in the color red, indicating that it is active.



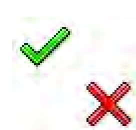
Introducing: Confirmation Corner

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** (upper right of the graphics window) 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.

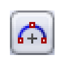
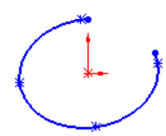
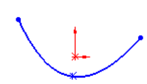
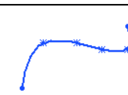
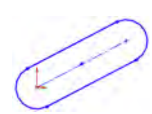
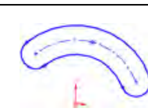


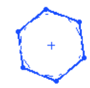
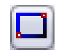
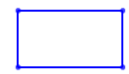
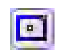
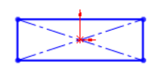
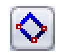
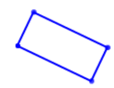
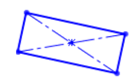
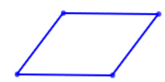
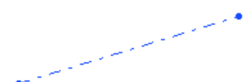
Sketch Entities

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 some of the sketch entities that are available on the Sketch toolbar.

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

Sketch Entity	Toolbar Button	Geometry Example
Polygon		
Corner Rectangle		
Center Rectangle		
3 Point Corner Rectangle		
3 Point Center 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. Additional clicks create a series of connected lines.
- **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 area, 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 42.

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5 Sketch a line.

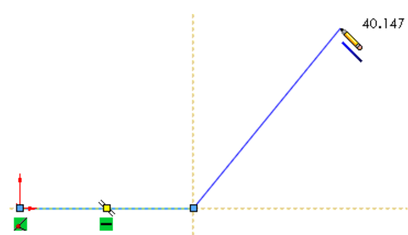
Click the **Line** tool and sketch a horizontal line from the origin. The “ ” symbol appears at the cursor, indicating that a **Horizontal** relation is automatically added to the line. The number indicates the length of the line. Click again to end the line.

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.

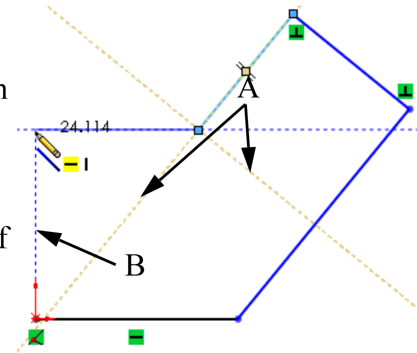
6 Line at angle.

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.

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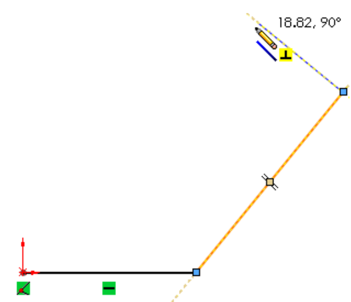
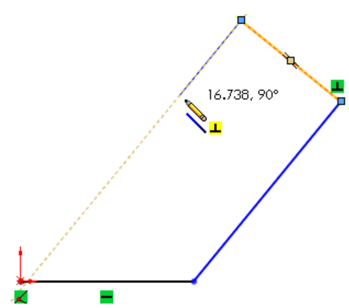
7 Inference lines.
Create a line moving in a direction perpendicular to the previous line. This causes inference lines to be displayed while sketching. A **Perpendicular** relation is created between this line and the last one.

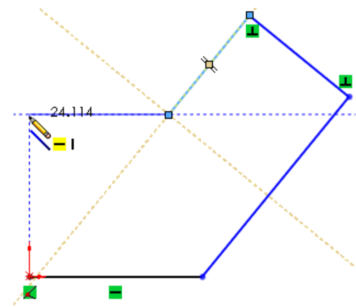


The cursor symbol indicates that you are capturing a perpendicular relation. Note that the line cursor is not shown for clarity.

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

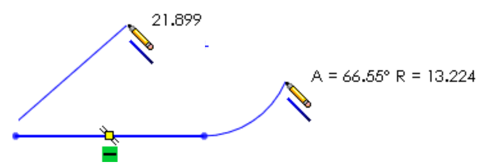


9 Reference.
Create a horizontal line from the last endpoint. Some inferences are strictly for reference and do *not* create relations. They are displayed in blue. This reference is used to align the endpoint vertically with the origin.

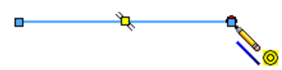
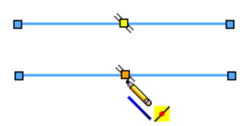
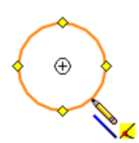


Sketch Feedback

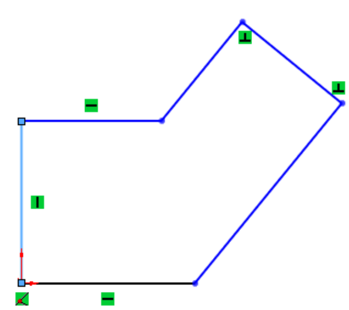
The sketcher has many feedback features. The cursor will change to show what type of entity is being created. It will also indicate what selections on the existing geometry, such as end, coincident (on) or midpoint, are available using a red dot when the cursor is on it.



Three of the most common feedback symbols are:

Symbol	Icon	Description
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.

10 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.
- Or, click the active tool (in this example **Line**) a second time.
- Or, click the **Select** tool.
- Or, right-click in the graphics area, and choose **Select** from the shortcut menu.

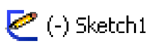
---

11 Turn off the tool.
Press the **Esc** key on the keyboard to turn off the line tool.

Status of a Sketch

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

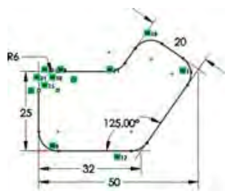
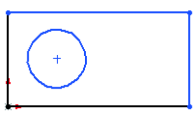
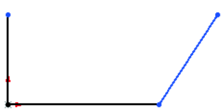
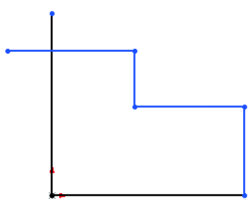
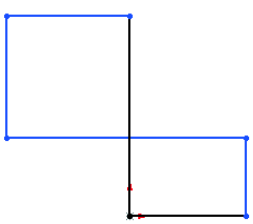
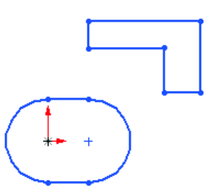
The sketch has duplicate dimensions or conflicting relations and it should not be used until repaired. Extraneous dimensions and relations should be deleted. Over defined geometry is **red** (by default).

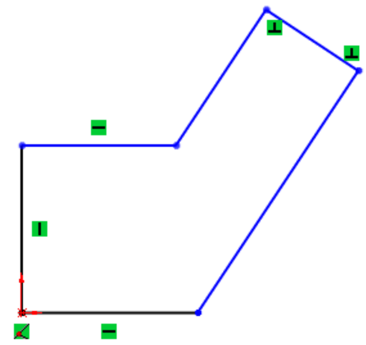
Note

The two other states are **No Solution Found** and **Invalid Solution Found**. They both indicate that there are errors that must be repaired. For more information on repairs, see *Lesson 8: Editing: Repairs*.

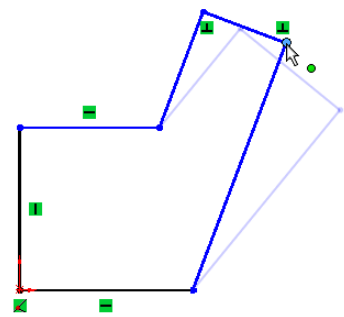
Rules That Govern Sketches

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 254.
	Corners are not neatly closed. *They should be.*	Use the **Contour Select Tool**. For more information, see *Sketch Contours* on page 319. 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 319. 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 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.



12 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.



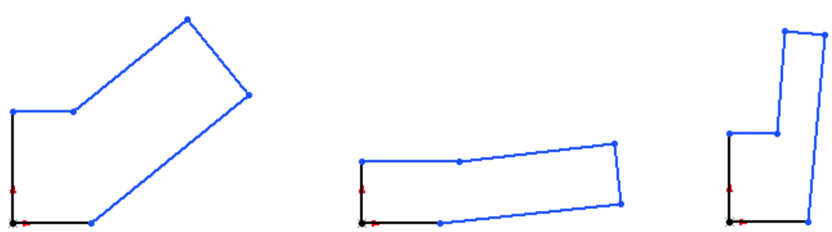
13 Dragging.
Under defined geometry (**blue**) can be dragged to new locations. Fully defined geometry cannot. Drag the uppermost endpoint to change the shape of the sketch. The dragged endpoint appears as a green dot.

14 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?

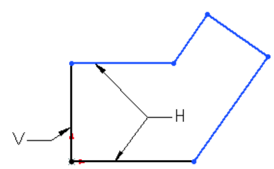
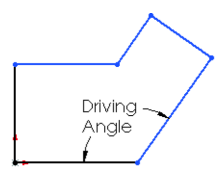
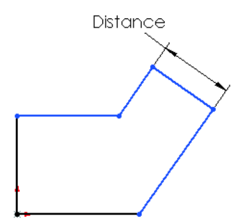
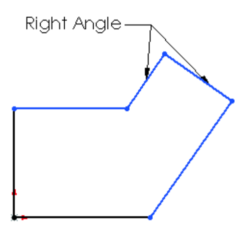
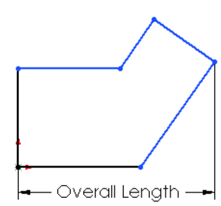
Design intent in a sketch is captured and controlled by a combination of two things:

- **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

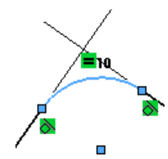
For those relations that cannot be added automatically, tools exist to create relations based on selected geometry.

Introducing: Display Relations

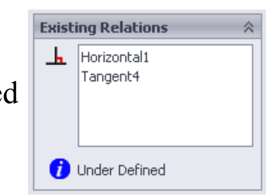
Display Relations shows and optionally enables you to remove geometric relationships between sketch elements.

Where to Find It

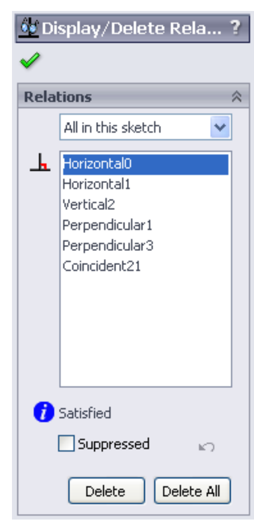
- Click the entity.
  Symbols appear indicating what relations are associated with that entity. In this example, the arc has three relations: two tangent and one equal.



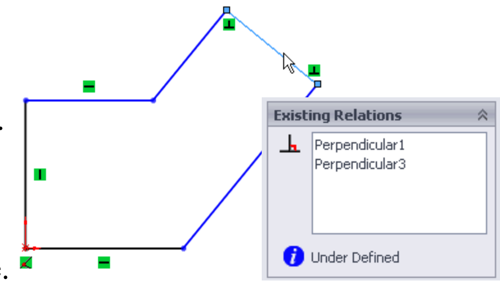
- The PropertyManager.
  Select the sketch entity and the PropertyManager shows the relations associated with that entity.



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



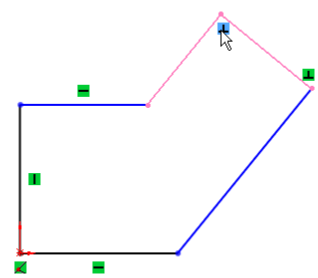
15 Display the relations associated with a line.
Click the uppermost angled line and the PropertyManager opens. The **Existing Relations** box in the PropertyManager also lists the geometric relations that are associated with the selected line.



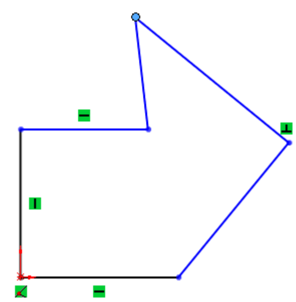
Tip

The relations are visible because **View, Sketch Relations** is turned on. If it is turned off, double-clicking the geometry will show the relations and open the PropertyManager.

16 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 changes color and displays the entitie(s) it controls.

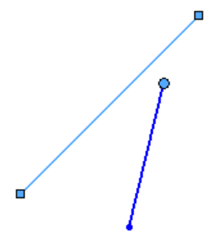
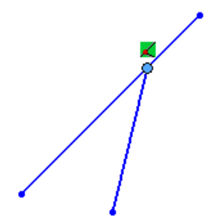
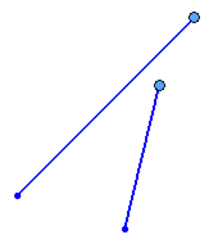
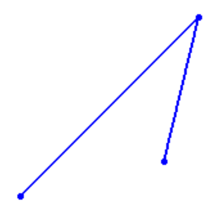
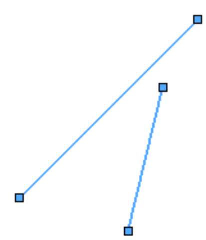
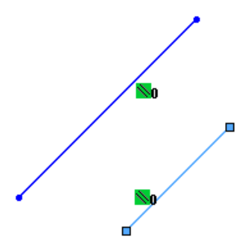
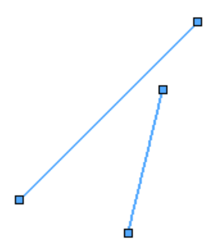
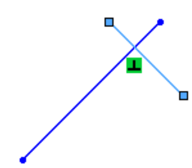


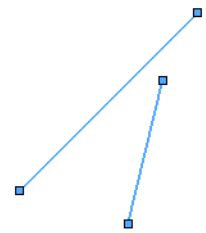
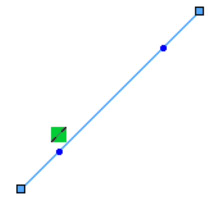
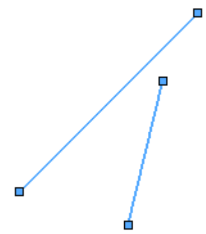
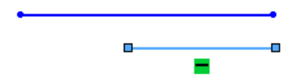
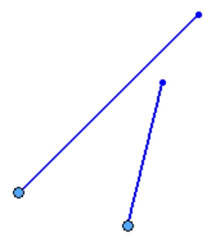
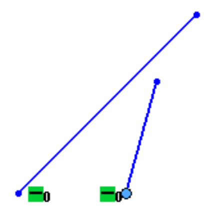
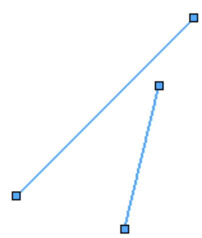
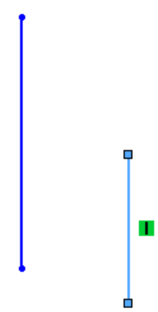
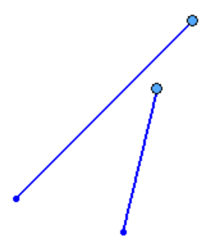
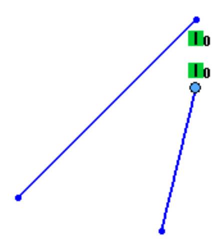
17 Drag the endpoint.
Because the line is no longer constrained to be perpendicular, the sketch will behave differently when you drag it. Compare this to how the sketch behaved when you dragged it in step **13**.

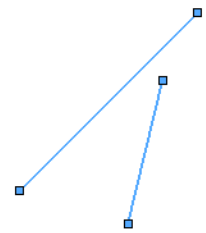
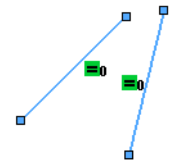
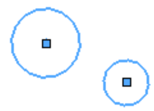
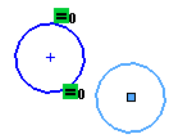
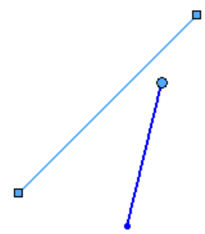
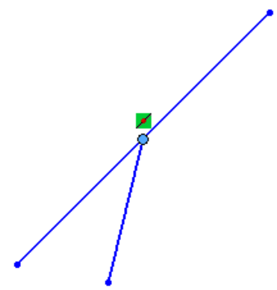


Examples of Sketch Relations

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 or more lines.		
Perpendicular between two lines.		

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

Relation	Before	After
Equal between two or more lines.		
Equal between two or more 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

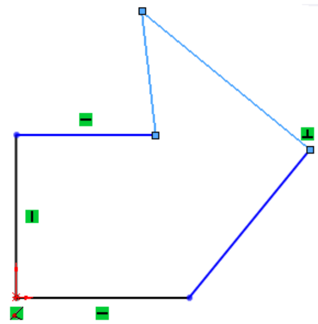
- Select the sketch entity or entities, and select the appropriate relation from the **Add Relations** section of the PropertyManager.
- Or, right-click the entity or entities, and select **Add Relation** from the shortcut menu.
- Or, click **Tools, Relations, Add**.
- Or, on the Sketch toolbar, click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

18 Add a relation.

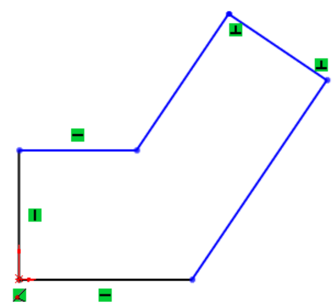
Hold down **Ctrl** and select the two lines. The PropertyManager shows only those relations that are valid for the geometry selected.



Click **Perpendicular**, and click **OK** or click in a blank area of the graphics window.

19 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.

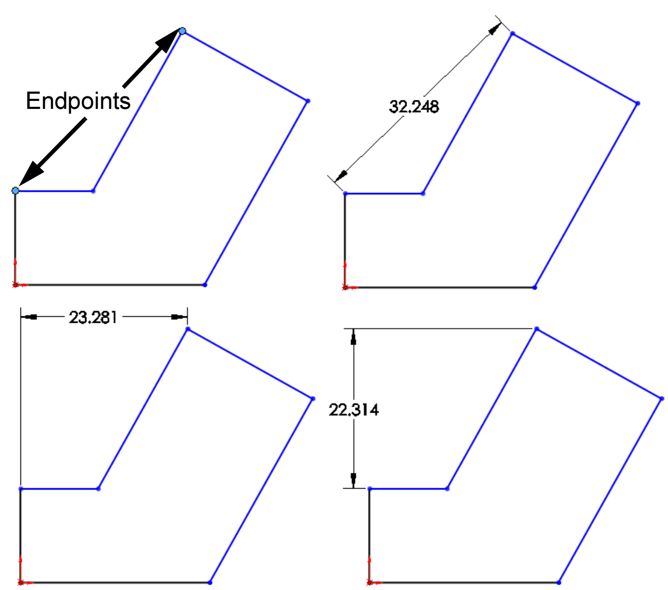
Where to Find It

- From the **Tools** menu, select **Dimensions, Smart**.
- Or, right-click off the geometry 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 enables 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.



Sketching with On Screen Numeric Input

An option to sketch and create dimensions at the same time is on screen numeric input. It requires these steps:

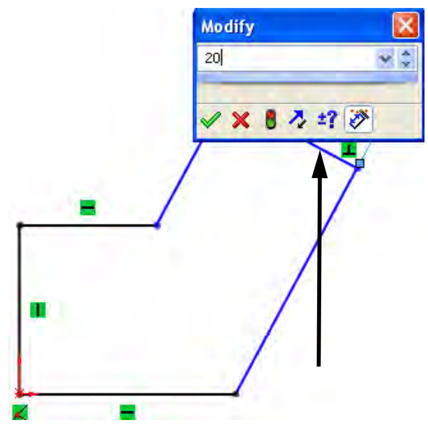
1. Click **Tools**, **Options**, **System Options**, **Sketch**, **Enable on screen numeric input on entity creation**.
2. **Add dimensions** on the PropertyManager of the selected sketch tool.
3. Use the sketch tool and type in values as they highlight.

Tip

At this early stage, it is inadvisable to use this option because it can inadvertently create an overdefined sketch (see *Status of a Sketch* on page 38).

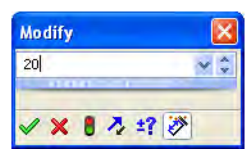
20 Adding a linear dimension.

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 thumbwheel is used to incrementally increase/decrease the value using the middle mouse button. 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:



 Dial 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.

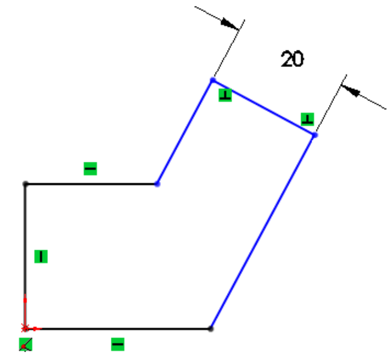
 Rebuild the model with the current value.

 Reverse the sense of the dimension.

 Change the thumbwheel increment value.

 Mark the dimension for drawing import.

21 Set the value.
Change the value to **20** and click the **Save** option. The dimension forces the length of the line to be 20mm.



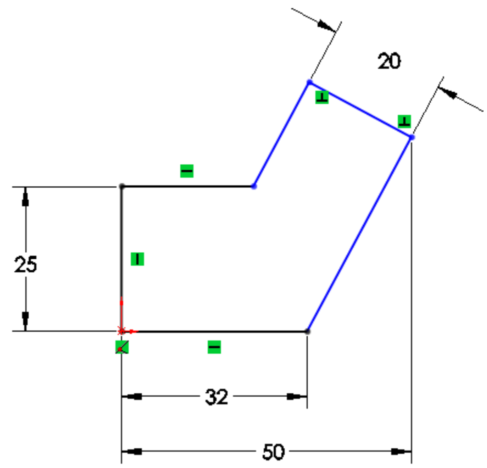
Tip

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

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

Dimensioning Tip

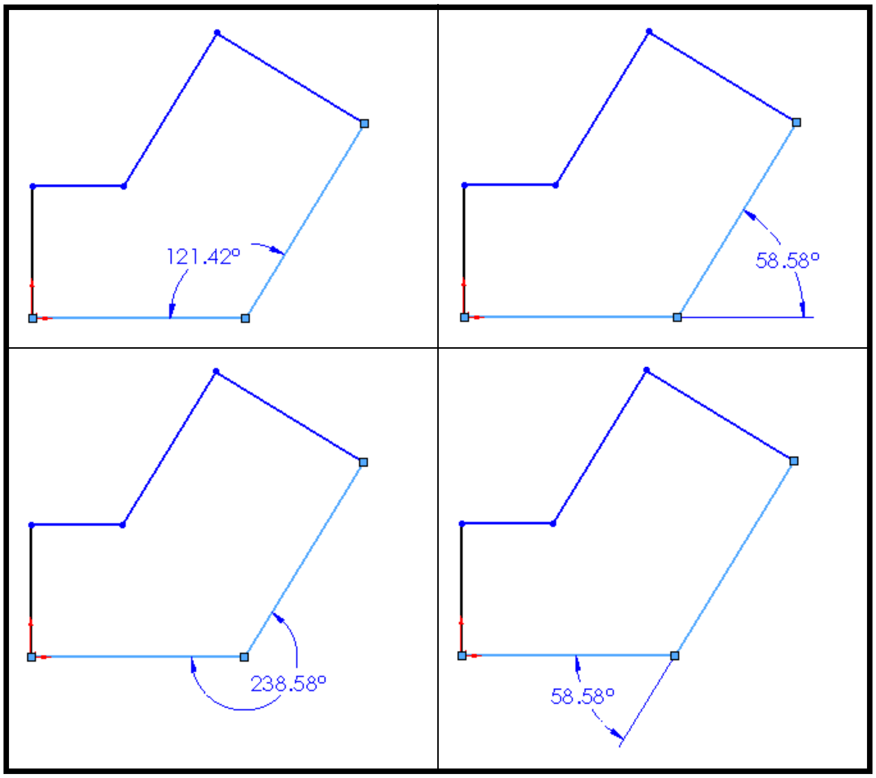
When you dimension a sketch, start with the smallest dimension first, and work your way to the largest.



Angular Dimensions

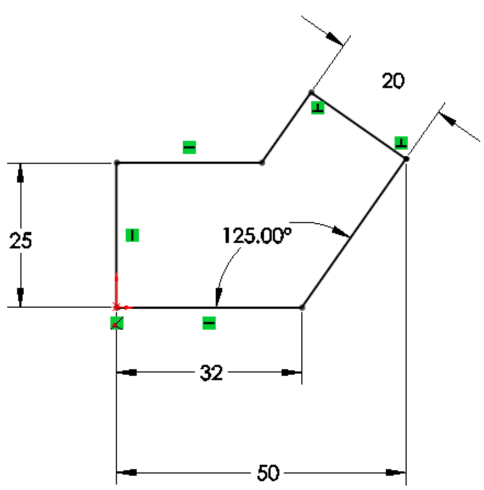
Angular dimensions can be created using the same dimension tool used to create linear, diameter and radial dimensions. Select either two lines that are both non-collinear and non-parallel, or select three non-collinear endpoints.

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:



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

The sketch is fully defined.



Extrude

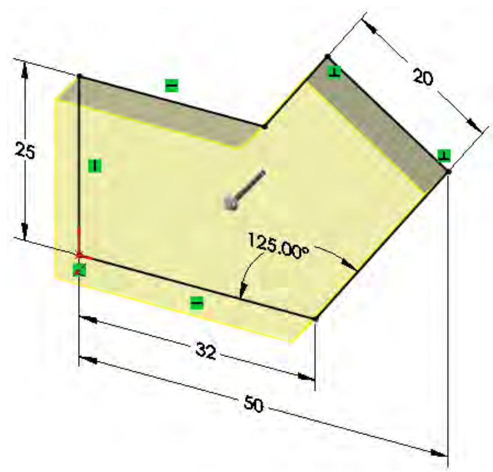
Once the sketch is completed, it can be extruded to create the first feature. There are many options for extruding a sketch including the start and end conditions, draft and depth of extrusion, which will be discussed in more detail in later lessons. Typically, extrusions take place in a direction normal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

24 Extrude menu.

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

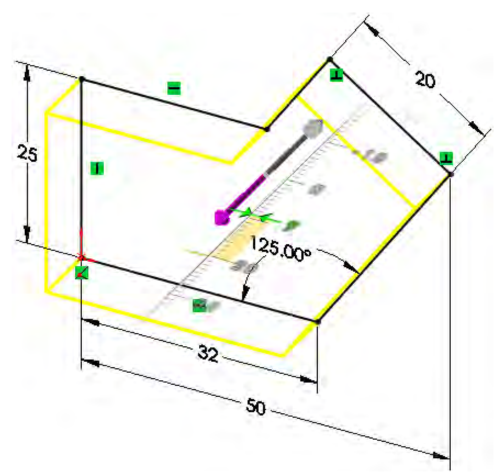


On the **Insert** menu, the options for other methods of creating features are listed along with **Extrude** and **Revolve**. They are unavailable because this sketch does not meet the conditions necessary for creating these types of features. For example, a **Sweep** feature requires both profile and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

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

Drag Handles and Rulers

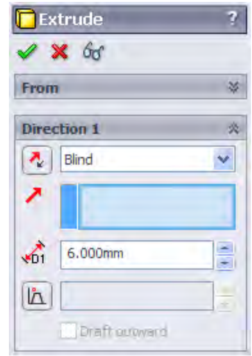
Handles appear that can be used to drag the preview to the desired depth. The handles are colored for the active direction and gray for inactive direction. A callout shows the current depth value and a **Ruler** is displayed to guide the drag. Moving closer to the ruler gradients allows you to snap to them.



Tip

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

25 Extrude Feature settings.
Change the settings as shown.



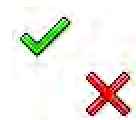
- **End Condition** = **Blind**
- (**Depth**) = **6mm**

Click **OK** to create the feature.

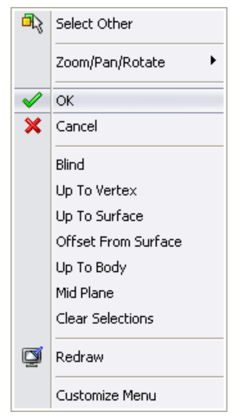
Tip

The **OK** button is just one way to accept and complete the process. A second is to press the **Enter** key.

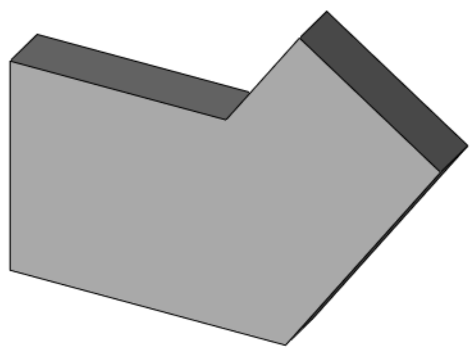
A third method is the set of **OK/Cancel** buttons in the **Confirmation Corner** of the graphics area.



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

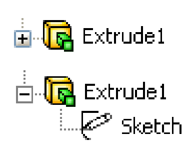


26 Completed feature.
The completed feature is the first solid, or feature of the part. The sketch is absorbed into the `Extrude1` feature.



Note

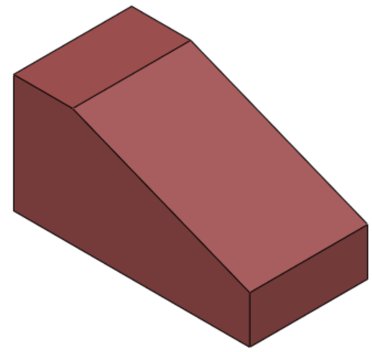
Click the ⊞ preceding the feature name to expand the feature, showing the absorbed feature, the sketch.



27 Save and close.
Click **Save** to save your work and click **File, Close** to close the part.

Exercise 1: Sketch and Extrude 1

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



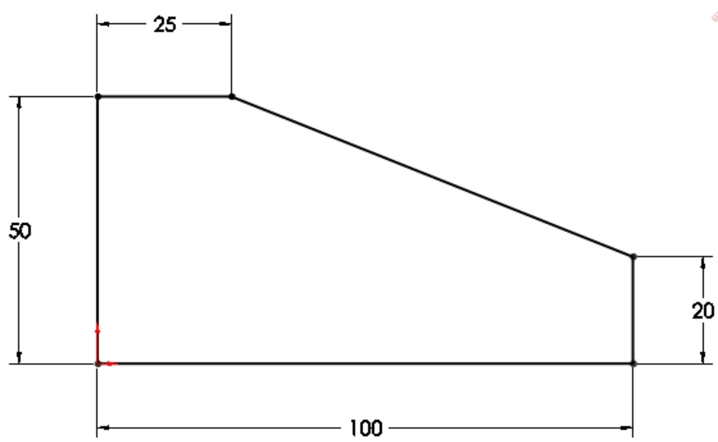
This lab reinforces the following skills:

- *Introducing: New Part* on page 29.
- *Sketching* on page 31.
- *Inference Lines (Automatic Relations)* on page 36.
- *Dimensions* on page 48.
- *Extrude* on page 52.

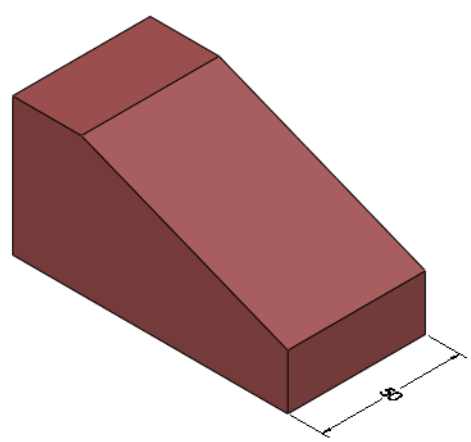
Units: **millimeters**

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

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



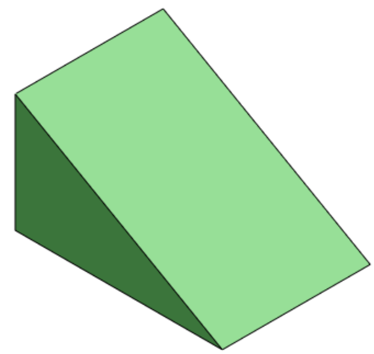
3 **Extrude.**
Extrude the sketch **50mm** in depth.



4 **Save and close the part.**

Exercise 2: Sketch and Extrude 2

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



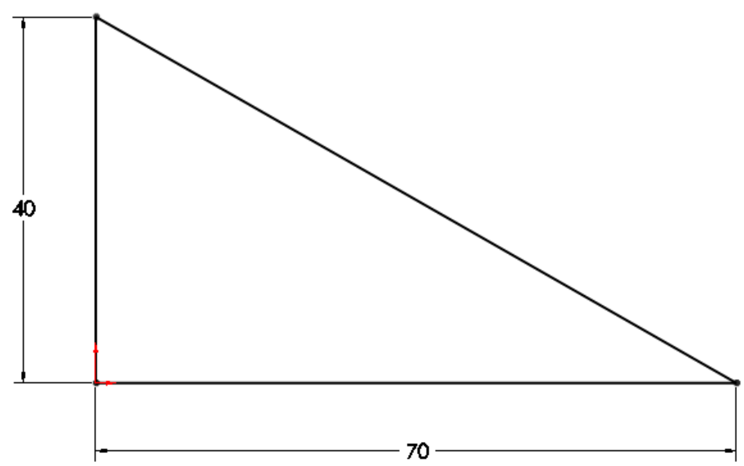
This lab reinforces the following skills:

- *Introducing: New Part* on page 29.
- *Sketching* on page 31.
- *Inference Lines (Automatic Relations)* on page 36.
- *Dimensions* on page 48.
- *Extrude* on page 52.

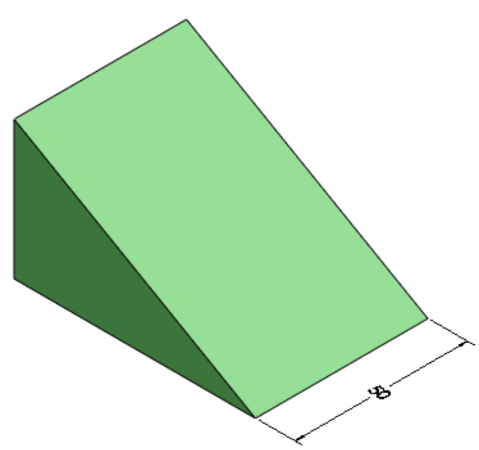
Units: **millimeters**

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

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



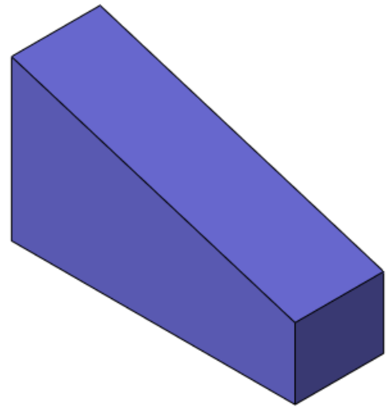
3 **Extrude.**
Extrude the sketch **50mm** in depth.



4 **Save and close the part.**

Exercise 3: Sketch and Extrude 3

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



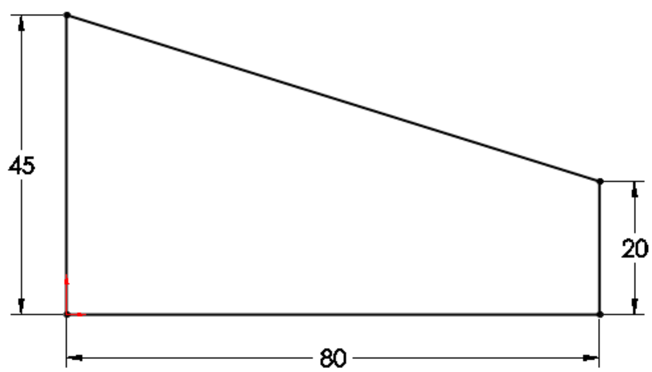
This lab reinforces the following skills:

- *Introducing: New Part* on page 29.
- *Sketching* on page 31.
- *Inference Lines (Automatic Relations)* on page 36.
- *Dimensions* on page 48.
- *Extrude* on page 52.

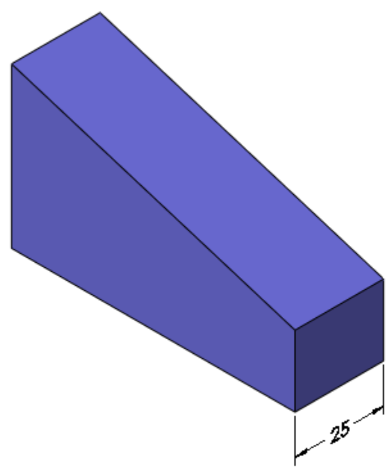
Units: **millimeters**

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

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



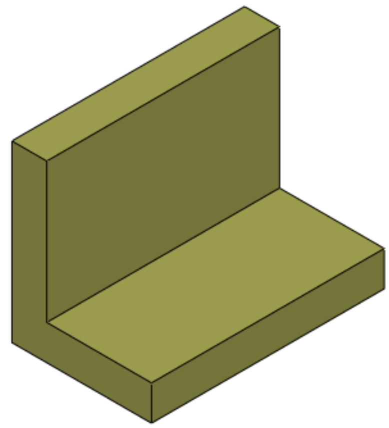
3 **Extrude.**
Extrude the sketch **25mm** in depth.



4 **Save and close the part.**

Exercise 4: Sketch and Extrude 4

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



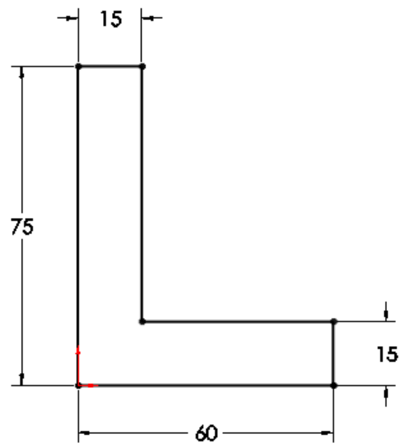
This lab reinforces the following skills:

- *Introducing: New Part* on page 29.
- *Sketching* on page 31.
- *Inference Lines (Automatic Relations)* on page 36.
- *Dimensions* on page 48.
- *Extrude* on page 52.

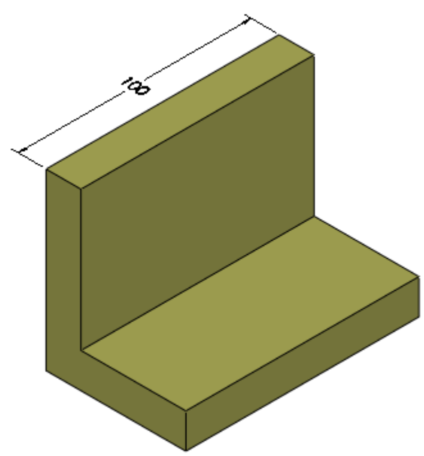
Units: **millimeters**

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

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



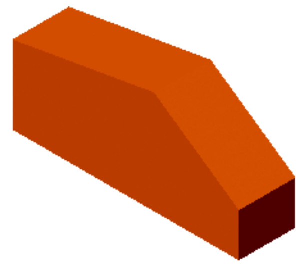
3 **Extrude.**
Extrude the sketch **100mm** in depth.



4 **Save and close the part.**

Exercise 5: Sketch and Extrude 5

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



This lab reinforces the following skills:

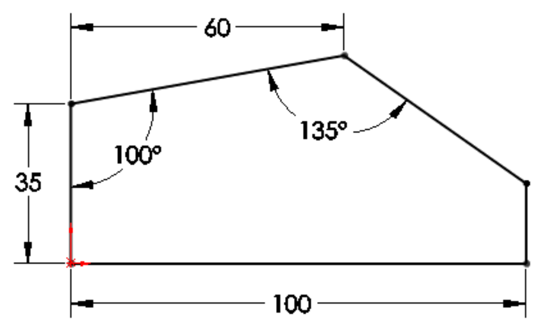
- *Introducing: New Part* on page 29.
- *Sketching* on page 31.
- *Inference Lines (Automatic Relations)* on page 36.
- *Dimensions* on page 48.
- *Extrude* on page 52.

Units: **millimeters**

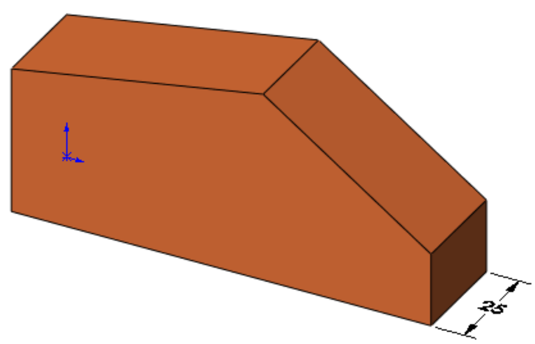
1 **New part.**
Open a new part using the `Part_MM` template.

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

Fully define the sketch.



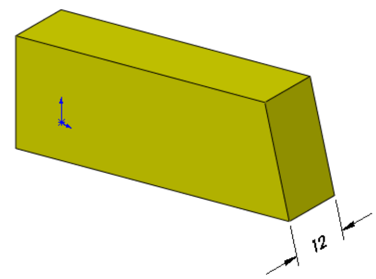
3 **Extrude.**
Extrude the sketch **25mm** in depth.



4 **Save and close the part.**

Exercise 6: Sketch and Extrude 6

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



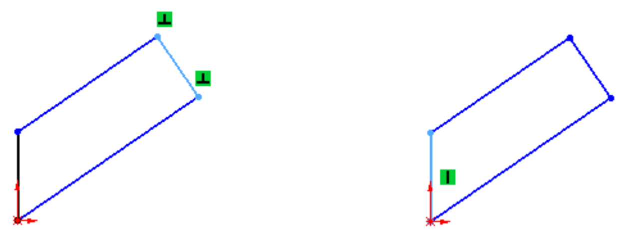
This lab reinforces the following skills:

- *Introducing: New Part* on page 29.
- *Sketching* on page 31.
- *Inference Lines (Automatic Relations)* on page 36.
- *Dimensions* on page 48.
- *Extrude* on page 52.

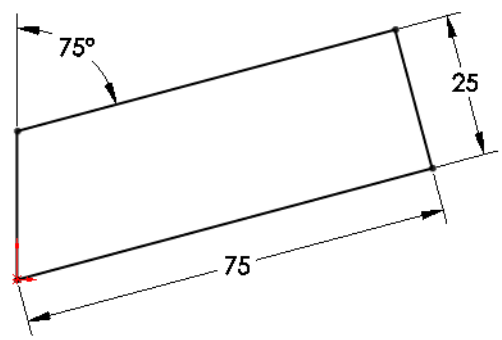
Units: **millimeters**

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

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

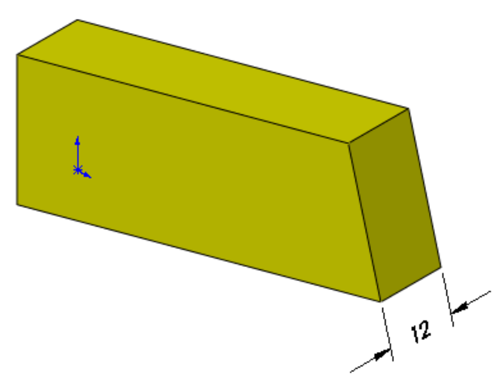


3 **Dimensions.**
Add dimensions to fully define the sketch.



4 **Extrude.**
Extrude the sketch **12mm**.

5 **Save and close the part.**



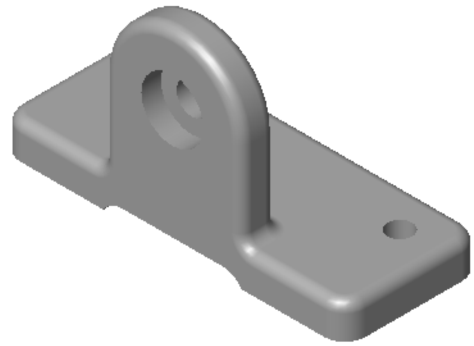
Lesson 3
Basic Part Modeling

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

- Choose the best profile for sketching.
- Choose the proper sketch plane.
- Extrude a sketch as a cut.
- Create Hole Wizard holes.
- Insert fillets on a solid.
- Use the editing tools edit sketch, edit feature and rollback.
- 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?
- **Bosses, cuts and hole features**
  How do you modify the first feature by adding bosses, cuts and holes?
- **Fillets**
  Rounding off the sharp corners – filleting.
- **Editing tools**
  Use three of the most common editing tools.
- **Drawings**
  Creating a drawing sheet and drawing views of the model.
- **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

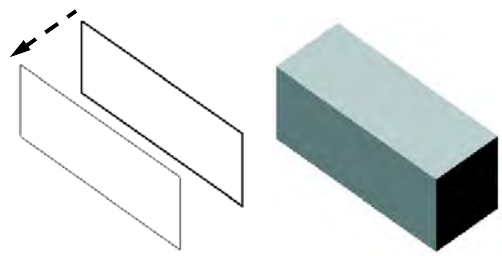
All cuts, bosses, planes and sketches that you create are considered Features. Sketched features are those based on sketches (boss and cut), and applied features are based on edges or faces (fillet).

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.

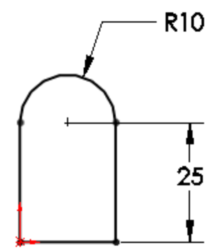
Extrusion

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 typically 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

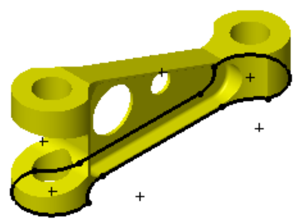
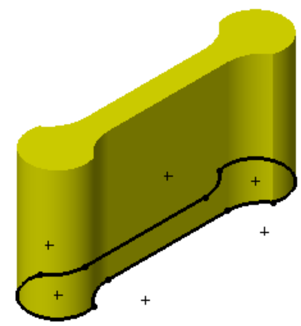
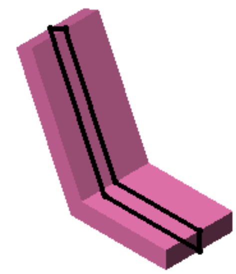
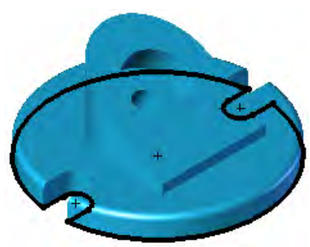
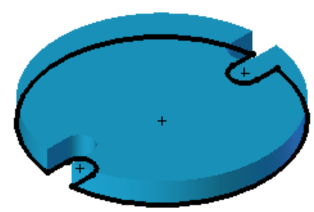
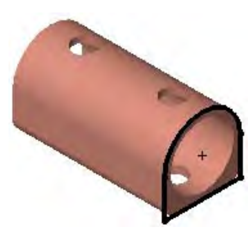
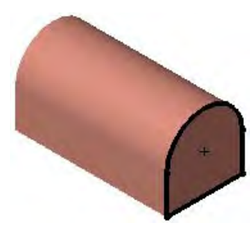
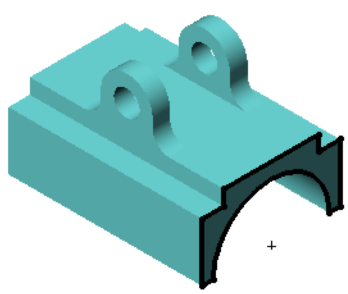
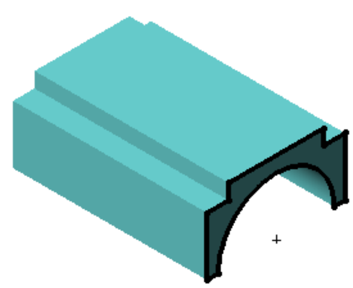
Fillets and *rounds* are generally added to the solid, not the sketch. By nature of the faces adjacent to the selected edge, the system knows whether to create a round (removing material) or a fillet (adding material).

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

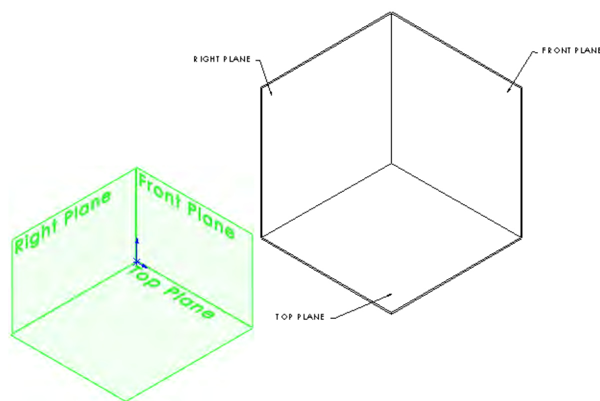
Once the best profile is determined, the next step is to decide which view to use and select the plane with the same name for sketching it. The SolidWorks software provides three planes; they are described below.

Planes

There are three default 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 Plane, Top Plane and Right Plane 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 the 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.

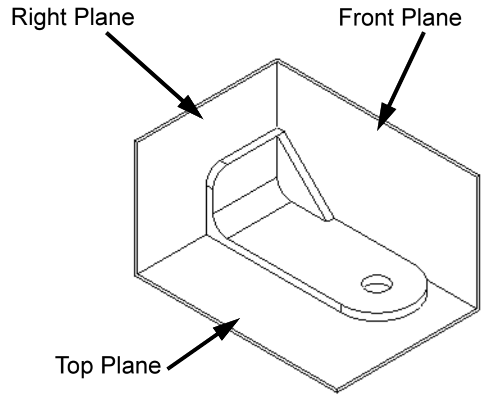
There are several things to consider when choosing the sketch plane. Two are appearance and the part's orientation in an assembly. The appearance dictates how the part will be oriented in standard views such as the Isometric. This also determines how you will spend most of your time looking at the model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

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

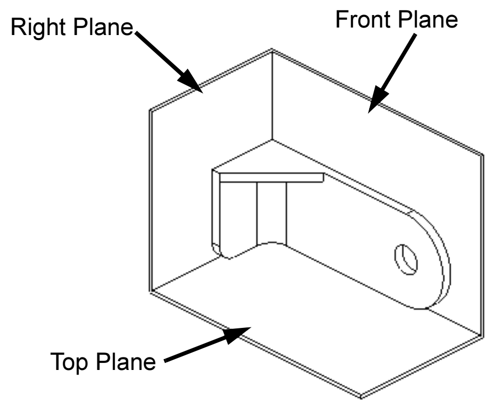
Orient the Model for the Drawing

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.

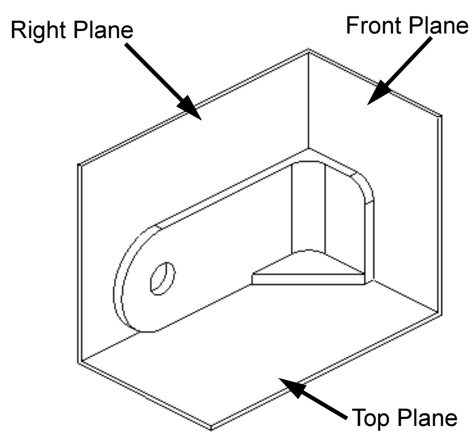
In the first example, the best profile is in contact with the Top plane.



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



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

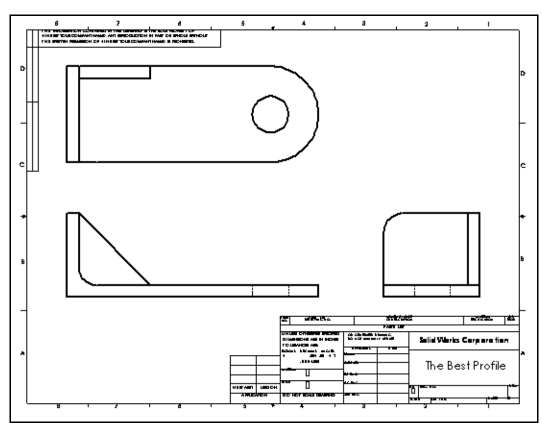


Chosen Plane

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.

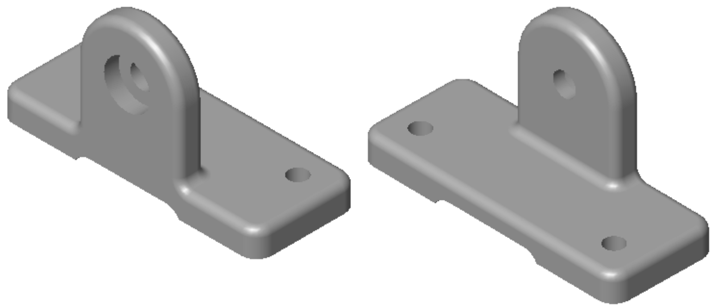
How it Looks on the Drawing

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



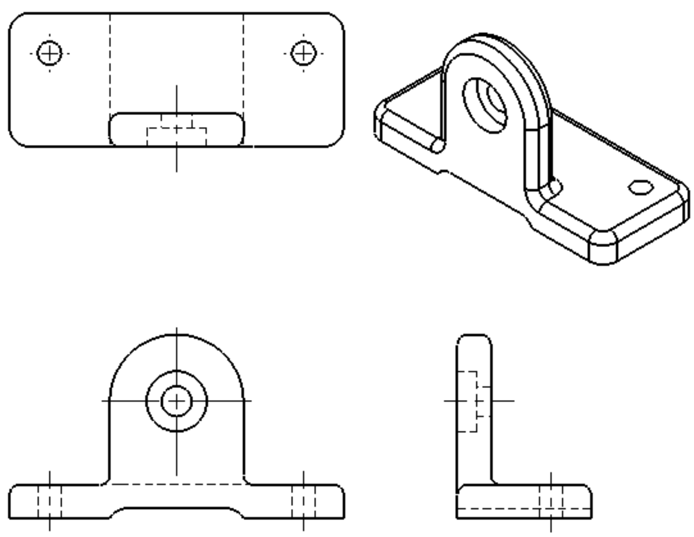
Details of the Part

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



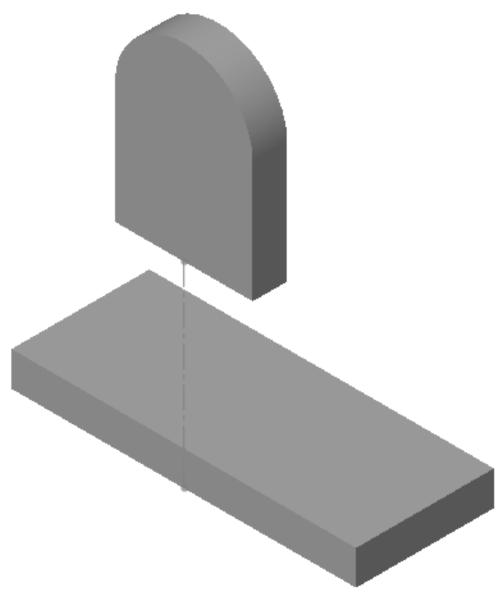
Standard Views

The part is shown here in four standard views.



Main Bosses

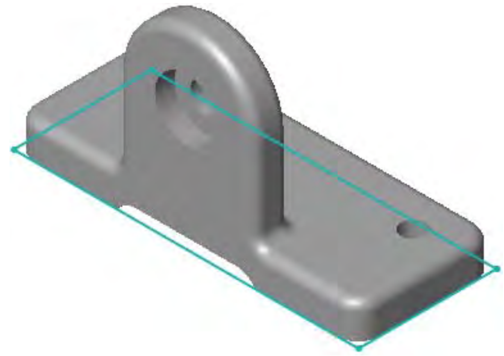
The two main bosses have distinct profiles in different planes. They are connected as shown in the exploded view at right.



Best Profile

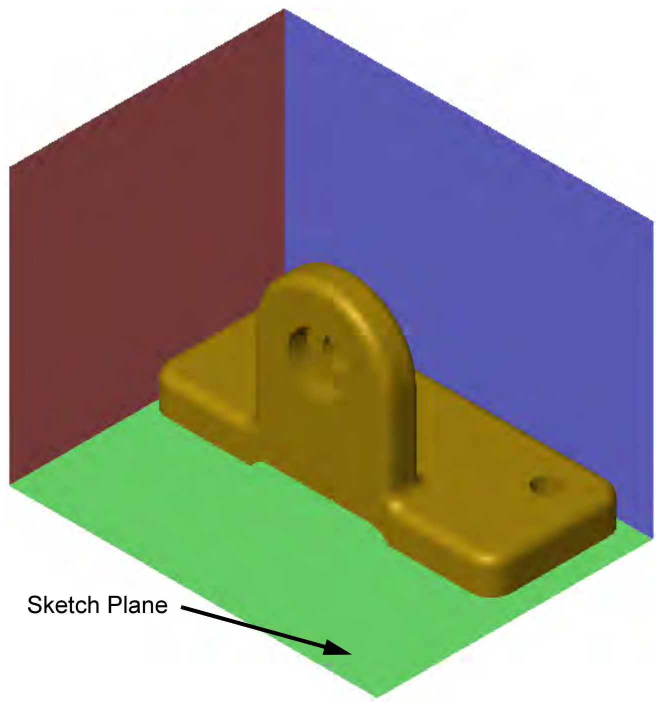
The first feature of the model is created from the rectangular sketch shown overlaid on the model. This is the best profile to begin the model.

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



Sketch Plane

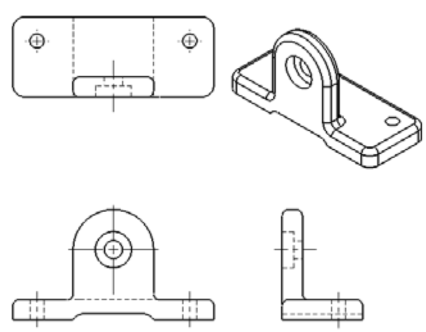
Placing the model "in the box" determines which plane should be used to sketch on. In this case it will be the Top plane.



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

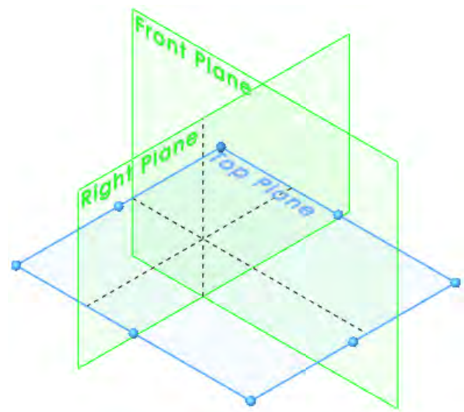
The modeling process includes sketching and creating bosses, cuts and fillets. To begin with, a new part file is created.

1 **New part.**

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

2 **Annotations setting.**
Right-click the Annotations folder and clear the **Automatically Place into Annotation Views** option. This will prevent dimensions from being inserted with drawing views later in the lesson.

3 **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. The first feature is always a boss, and it is the first solid feature created in any part. Begin with the sketch geometry, a rectangle.

Introducing: Corner Rectangle

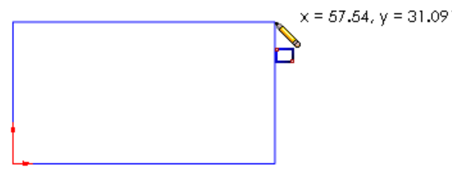
Corner Rectangle is used to create a rectangle in a sketch. The rectangle is comprised of four lines (two horizontal and two vertical) connected at the corners. It is sketched by indicating the locations of two diagonal corners. There are several other rectangle/parallelogram tools available:

- **Center Rectangle** - Uses a center point and corner to create a rectangle with horizontal and vertical lines.
- **3 Point Center Rectangle** - Creates a rectangle based on a center point, midpoint of edge and corner. Lines are perpendicular at corners.
- **3 Point Corner Rectangle** - Uses three corners to define a rectangle. Lines are perpendicular at corners.
- **Parallelogram** - Uses three corners to define a *parallelogram* (corners are not perpendicular).

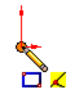
Where to Find It

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

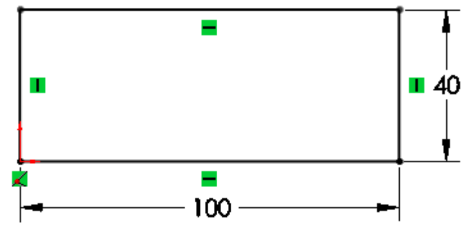
4 Sketch a rectangle.
Click the **Corner Rectangle** tool and begin the rectangle at the origin.



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.



5 Fully defined sketch.
Add dimensions to the sketch. The sketch is fully defined.



Extrude Options

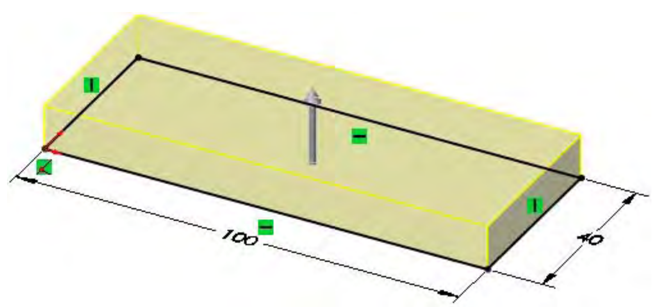
An explanation of some of the more frequently used **Extrude** options is given below. Other options will be discussed in later lessons.

- **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.

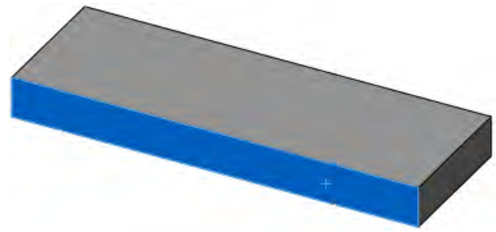
6 Extrude.

Click **Extrude** and extrude the rectangle **10mm** upwards.

Click **OK**.



The completed feature is shown at the right.



Renaming Features

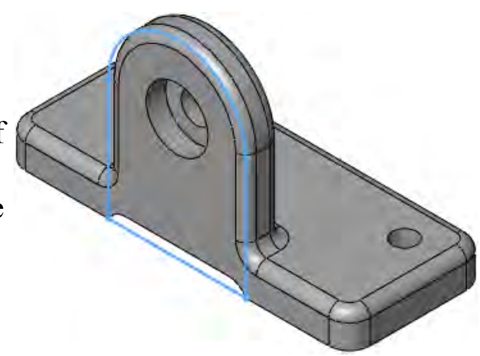
Any feature that appears in the FeatureManager design tree (aside from the part itself) can be renamed using the procedure below. Renaming features is a useful technique for finding and editing features in later stages of the model. Well chosen, logical names help you to organize your work and make it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

7 **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 plane, but a planar face of the model. The required sketch geometry is shown overlaid on the finished model.

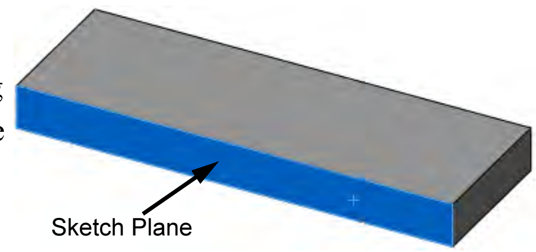


Tip Cut features are created in the same way as bosses – with a sketch and extrusion. They remove material rather than add it.

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.

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



Note

Make sure that **Instant 3D** (Features toolbar) is turned off. Leaving it on will cause several handles and axes that we are not currently using to appear on the 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

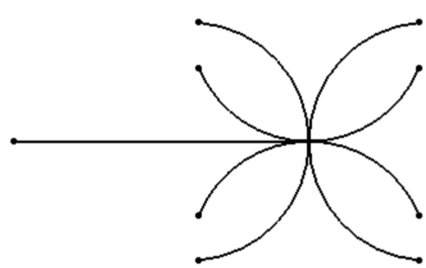
Insert Tangent Arc is used to create tangent arcs in a sketch. The arc must be tangent to some other entity, line or arc, at its start.

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.

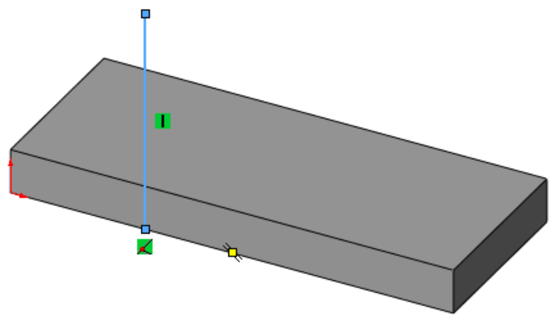
- A preview shows what type of arc you are sketching.
- You can change from one to the other by returning the cursor to the endpoint and moving away in a different direction.

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 returning the cursor to the endpoint and moving away in a different direction or by pressing the **A** key on the keyboard.

9 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.



10 Autotransition.

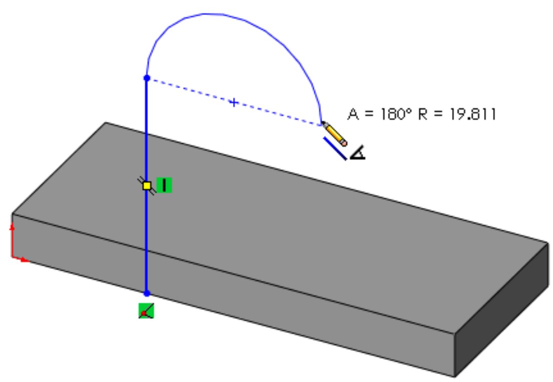
Press the letter **A** on the keyboard.

You are now in tangent arc mode.

11 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.

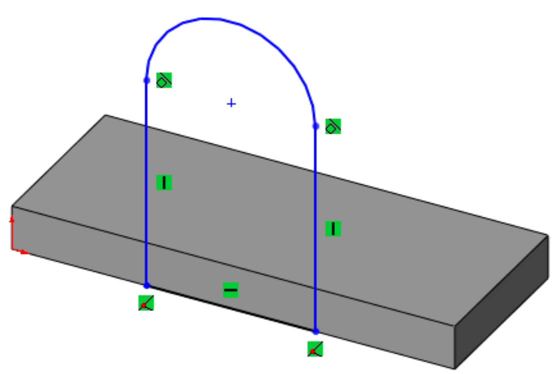
When you finish sketching the tangent arc, the sketch tool automatically switches back to the line tool.



12 Finishing lines.

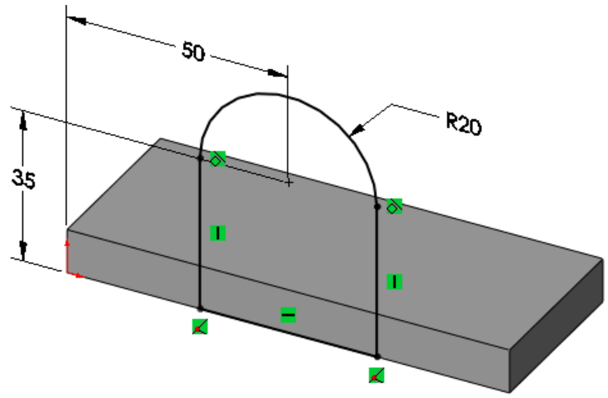
Create a vertical line from the arc end to the base, and one more line connecting the bottom ends of the two vertical lines.

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



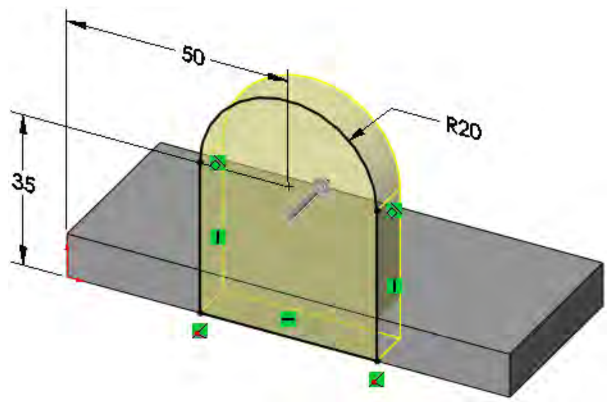
13 Add dimensions.
Add linear and radial dimensions to the sketch.

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



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

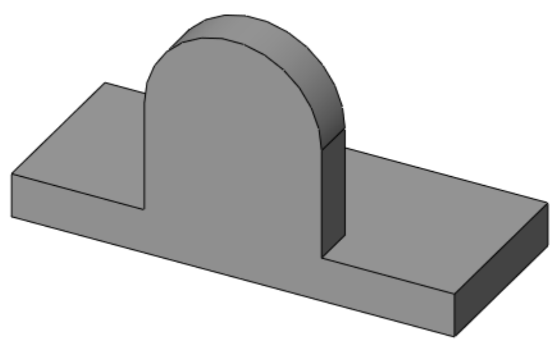
14 Extrude direction.
Click **Insert, Boss, Extrude** and set the **Depth** to **10mm**. Note that the preview shows the extrusion going into the base, in the proper direction.



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

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

Rename the feature VertBoss.



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.

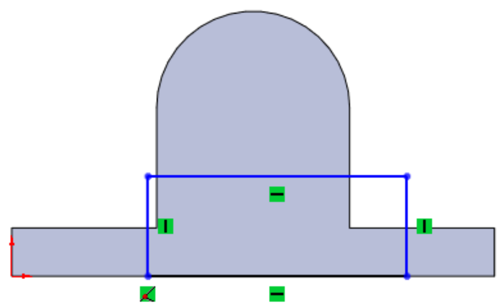
Where to Find It

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

---

16 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.

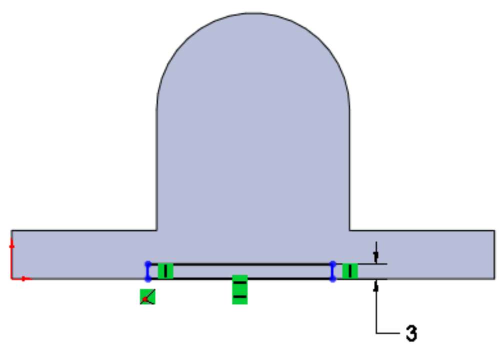
Turn off the rectangle tool.



Selecting Multiple Objects

As you learned in *Lesson 2* on page 27, when selecting multiple objects, hold down the **Ctrl** key and then select the objects.

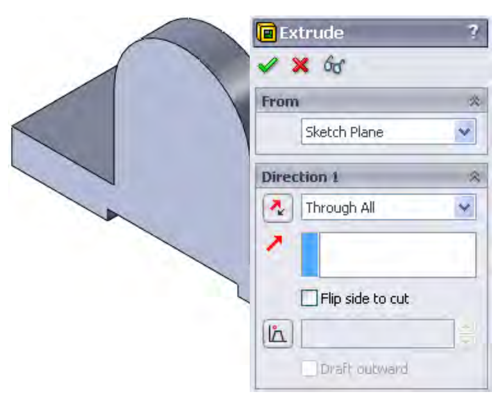
17 Relations.
Add a dimension as shown. Change the view orientation to Isometric.



Note

The sketch is under defined. See *Status of a Sketch* on page 38

18 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.



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 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 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 or sketch to be selected or pre-selected.

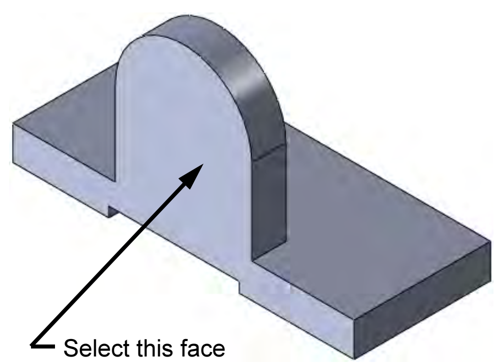
Where to Find It

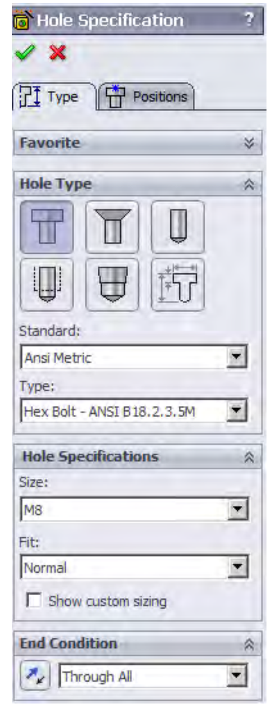
- From the **Insert** menu, choose **Features, Hole, Wizard...**.
- Or, choose the **Hole Wizard** tool on the Features toolbar.

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.

19 Hole position.
Select the face indicated and **Insert, Features, Hole, Wizard...**.





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

Type: Counterbore

Standard: Ansi Metric

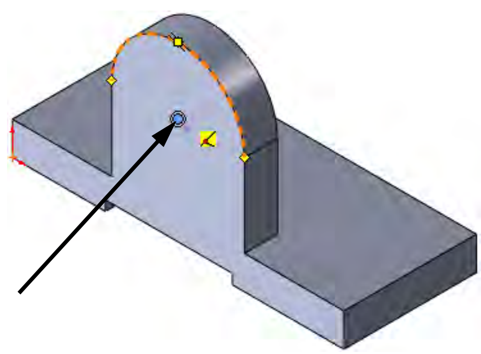
Type: Hex Bolt

Size: M8

End Condition: Through All

Click the **Positions** tab.

21 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.

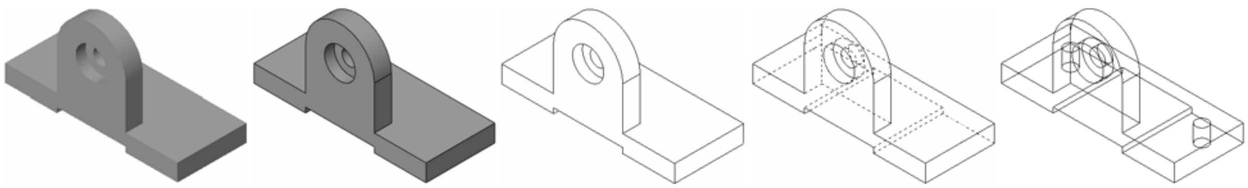
Drop the point onto the arc's centerpoint. Look for the feedback that tells you that you are snapping to the arc's center, a coincident relation. Click **OK** to add the relation and again to complete the dialog.

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 (adding volume) and rounds (removing volume). 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.

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.

Tip

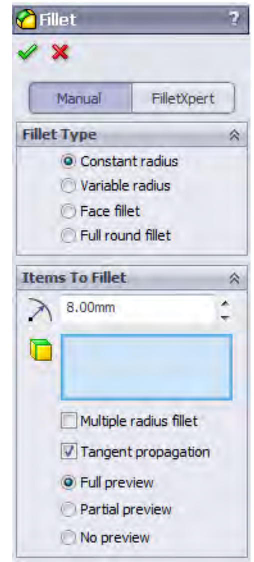
The *FeatureXpert* on page 280 can be used to automate the sizing and ordering of fillets. It will be discussed in *Lesson 8: Basic Part Modeling*.

Where to Find It

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

22 Insert Fillet.

Select the **Fillet** option in one of the ways mentioned above. The **Fillet** options appear in the PropertyManager. Click **Manual** and set the radius value.



- (**Radius**) = **8mm**

Preview

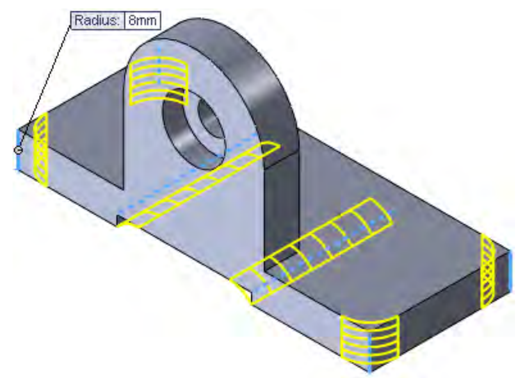
You have a choice between **Full preview**, **Partial preview** and **No preview** of the fillet. **Full preview**, as shown below, generates a mesh preview on each selected edge. **Partial preview** only generates the preview on the first edge you select. As you gain experience with filleting, you will 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 (for **Fillet** and **Chamfer** only).

23 Edge selection.

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



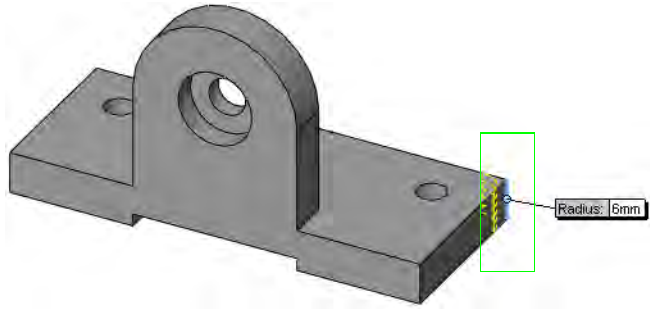
A callout Radius: 6mm appears on the first edge you select. Select six edges total and click **OK**.

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.

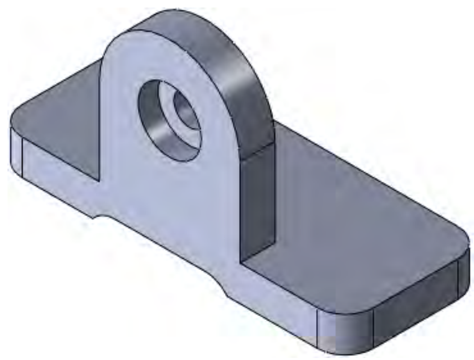
Tip

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.



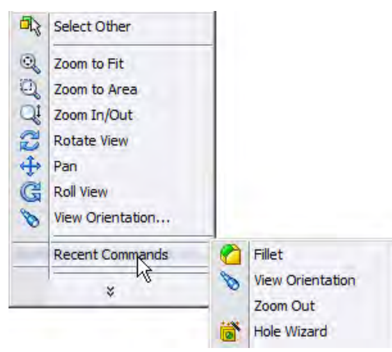
24 Completed fillets.
All six fillets are controlled by the same dimension value. The creation of these fillets has generated new edges suitable for the next series of fillets.

Return to the **Isometric** view.



Recent Commands Menu

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



25 Recent Command.
Right-click in the graphics area and select **Recent Commands** and the **Fillet** command from the drop-down list to use it again.

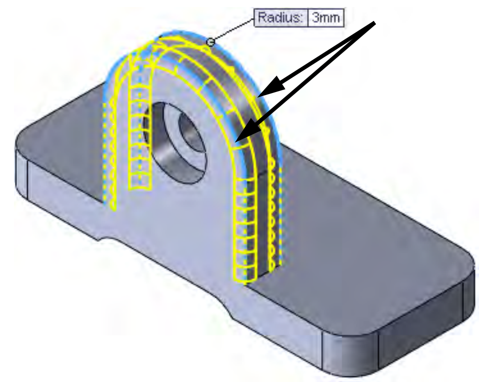
Fillet Propagation

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

26 Preview and propogate.

Add another fillet, radius **3mm**, using **Full preview**.

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



Editing Tools

Three of the most useful editing tools are introduced in this lesson: **Edit Sketch**, **Edit Feature** and **Rollback**. They can be used to edit and repair sketches and features as well as specify where, in the FeatureManager design tree, the features are to be created.

Tip

The other editing tools are found later in this lesson: *Editing Features* on page 82 and *Rollback* on page 83.

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 enables 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.

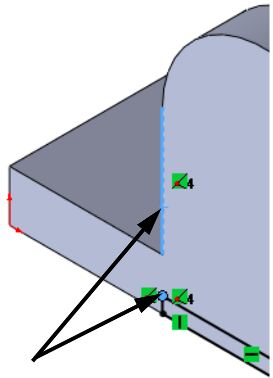
Where to Find It

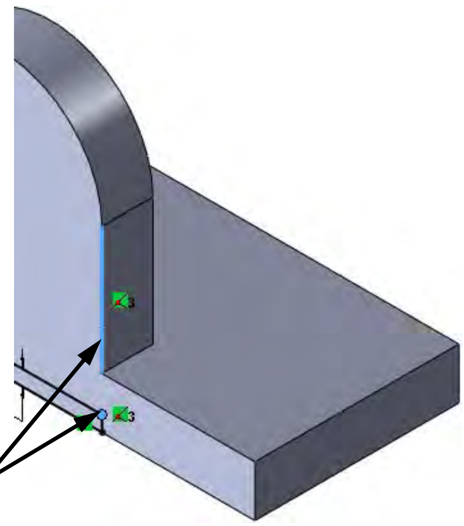
- From the **Edit** menu, choose **Sketch**.
- Or, click or right-click the feature whose sketch you want to edit and select **Edit Sketch**.

27 Edit the sketch.

Right-click the `BottomSlot` feature and select **Edit Sketch**. The existing sketch will be opened for editing.

28 Relations.
Select the endpoint and edge as shown and add a Coincident relation.





Repeat the procedure for the set on the opposite side as shown. The addition of these relations will fully define the sketch.

Note

For more information about relations, see *Sketch Relations* on page 42.

29 Exit the sketch.

Click **Exit Sketch** in the upper right (confirmation) corner to exit the sketch and rebuild the part.

Editing Features

The second fillet should propagate into the larger diameter fillet. To do this we will edit the definition of the `last fillet feature`.

Introducing: Edit Feature

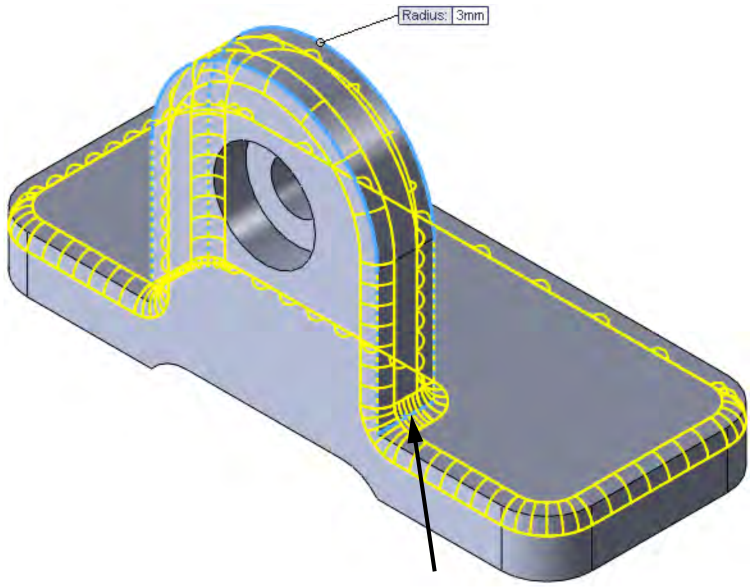
Edit Feature changes how a feature is applied to the model. Each feature has specific information that can be changed or added to, depending on the type of feature it is. As a general rule, the same dialog box used to create a feature is used to edit it.

Where to Find It

- Click or right-click the feature to edit – either in the FeatureManager design tree or the graphics area, and select the **Edit Feature** icon.

30 Edit the feature.
Right-click the `Fillet2` feature and select **Edit Feature**. The existing feature will be opened for editing using the same PropertyManager that was used to create the feature.

31 Select additional edge.
Select the additional edge as shown and the propagation will create the fillets as shown. Click **OK**.



Rollback

The **Rollback** bar is the blue horizontal bar located at the bottom of the FeatureManager design tree.

It is a tool that has many uses. It can be used to "walk through" a model showing the steps that were followed to build it or to add features at a specific point in the part's history. In this example, it will be used to add a hole feature between the existing fillet features.

Introducing: The Rollback Bar

You can roll back a part using the **Rollback Bar** in the FeatureManager design tree. The rollback bar is a line which highlights 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 previous 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.

Tip

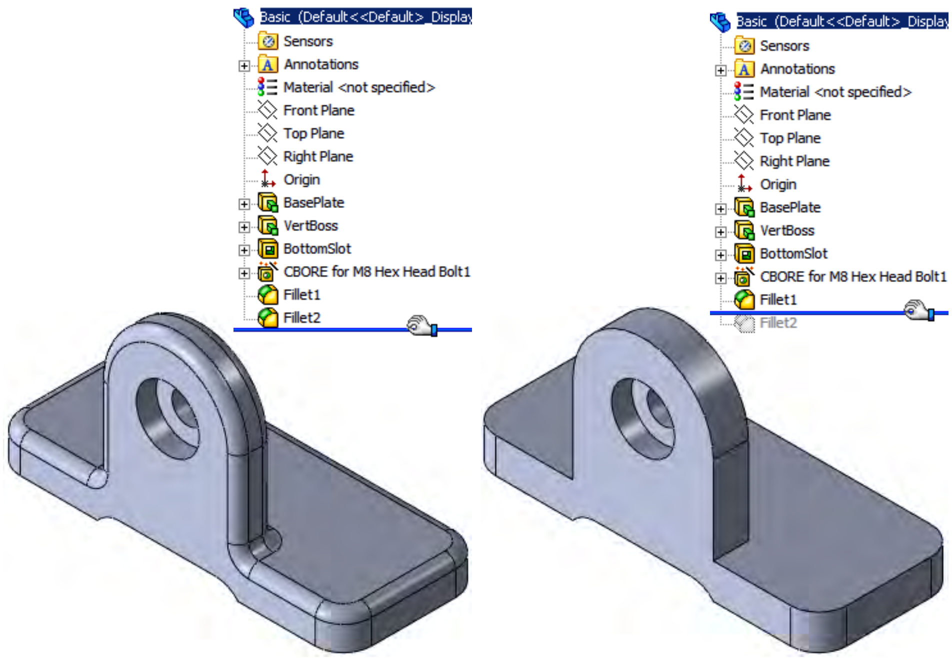
The focus must be set to the rollback bar by clicking on it. If the focus is set to the graphics area, the arrow keys will rotate the model.

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. The part can be saved in a rollback state.

---

32 Rollback.
Click on the **Rollback** bar and drag it upwards. Drop it between the fillet features as shown.



33 Hole wizard.

Click the **Hole Wizard** tool and click the **Positions** tab.

34 Select face.
Select the top, flat face of the base feature near the location shown.

Tip

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

35 A second hole.

Float over the arc edge to “wake up” the centerpoint. Place the point at the centerpoint.

For more information, see step **21** on page 77.

36 Move first hole.

Using the same procedure, drag the first point to the centerpoint on the opposite side.

37 Type.

Click the **Type** tab. Set the properties of the hole as follows:

Type: Hole

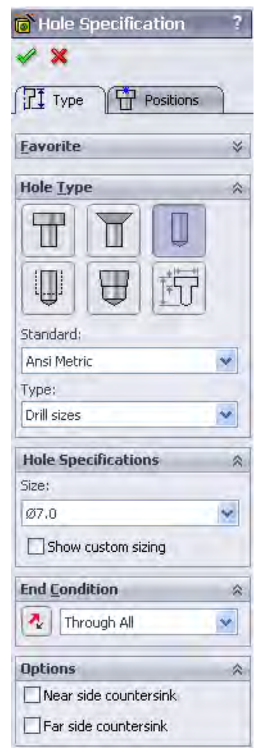
Standard: Ansi Metric

Type: Drill sizes

Size: 7.0

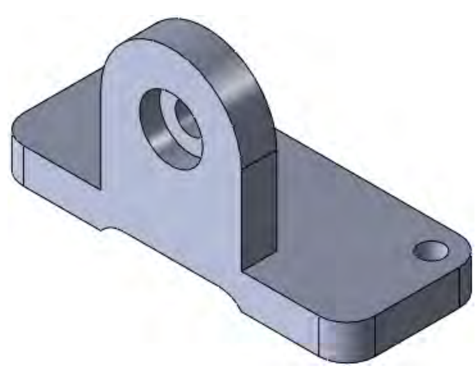
End Condition: Through All

Click **OK**.

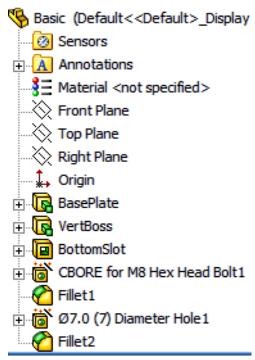


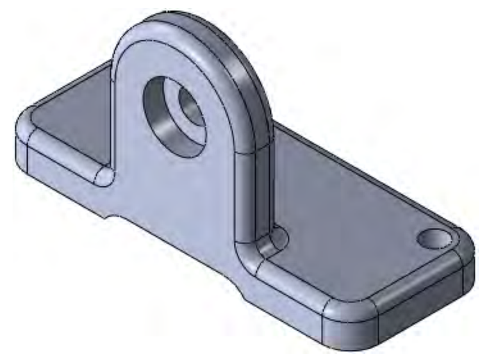
38 Change the view orientation.

Click **Isometric** to change view orientation.



39 Roll to end.
Click on the rollback bar and right-click **Roll To End**.





Introducing: Appearances

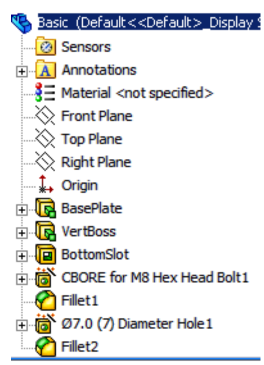
Use **Appearances** to change the color and optical properties of graphics. Color **Swatches** can also be created for user defined colors.

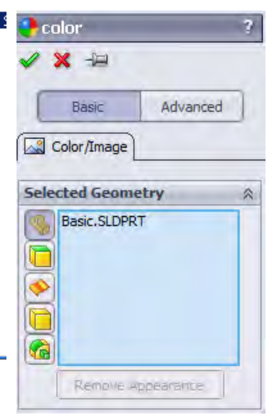
Where to Find It

- Right-click the top level feature (part name) and choose **Appearances** and the part name.
- Or, click the part and choose **Appearances** followed by face, feature, body, or the part.

---

40 Appearance.
Right-click the top level feature and choose **Appearances** and the part name Basic.

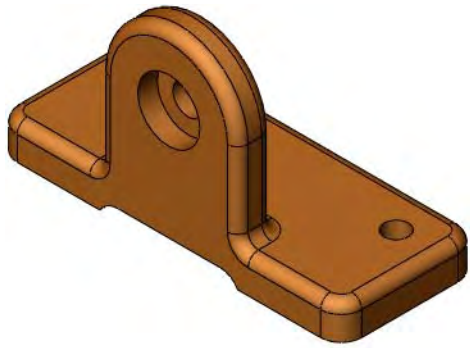




41 Select Swatch.

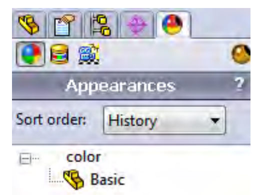
Under the **Color** selection, select the `standard` swatch and one of the colors as shown.

Click **OK**.



42 Display appearances.

Click the **Display Manager** tab to see the color listed. Click the FeatureManager design tree tab.



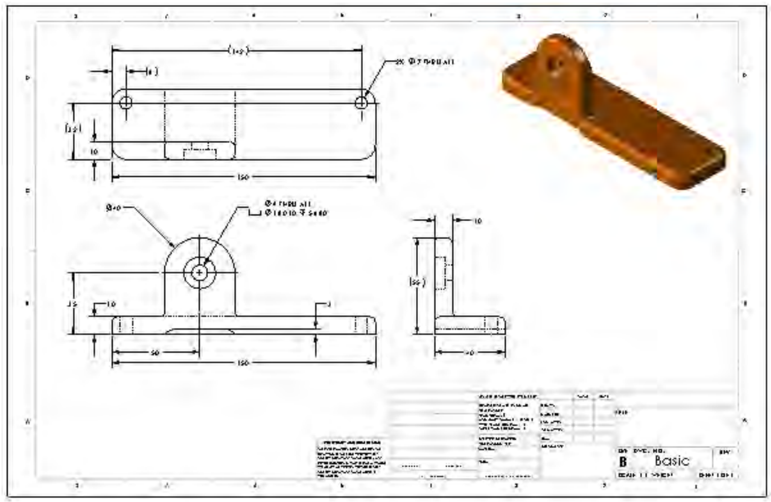
Tip The **Display Manager** can also be used to view decals, scenes, lights and cameras.

43 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 drawing views using the View Palette.
- Using dimension assist tools.

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

Settings Used in the Template

The drawing template used in this section has been designed to include the **Document Properties** shown in the chart below. Settings are accessed through **Tools, Options**. The settings that will be 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**	Drafting Standard: • **Overall drafting standard = ANSI**
	Tables: **Bill of Materials, Automatic update of BOM = Selected**
Colors: • **Drawings, Hidden Model Edges = Black**	Dimensions: • **Font = Century Gothic** • **Primary precision = .123** • **Add parentheses by default = Selected**
	Detailing, Auto insert on view creation: • **All options = cleared**
	Units • **Unit system =MMGS**

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 **File**, **Make Drawing from Part/Assembly** icon and choose `B_Size_ANSI_MM` from the **Training Templates** tab.

The sheet format creates a B-size drawing (11” x 17”) arranged with its long edge horizontal. The sheet format includes a border, title block, and other graphics.

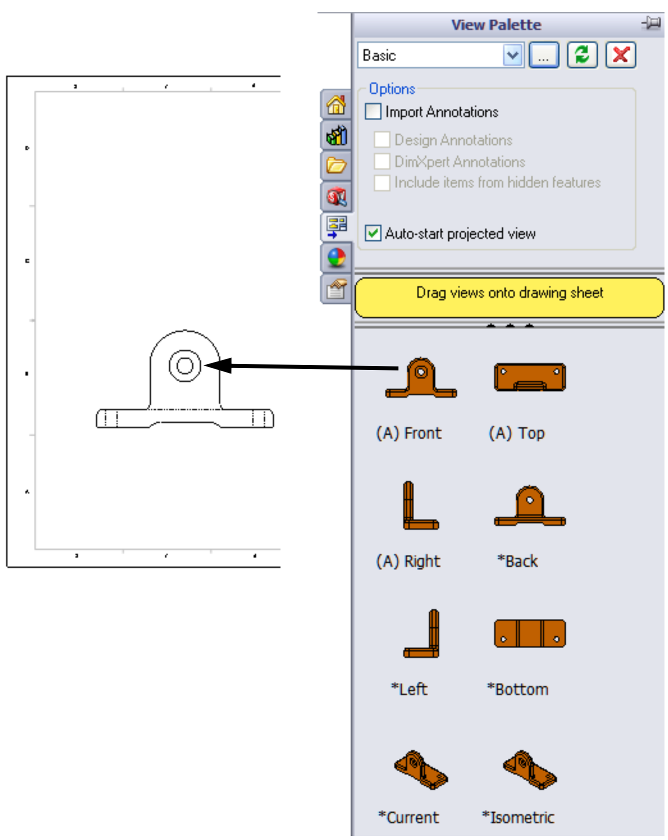
Tip Double-clicking the template will automatically open it, eliminating the need to click **OK**.

Drawing Views

The initial task of detailing is the creation of views. Using the **Make Drawing from Part/Assembly** tool leads you through the selection of the drawing sheet to the **View Palette**. This option generates drawing views that match the orientations in the part using a drag and drop procedure. Additional views can be projected or folded directly from the dropped view.

These options are discussed in detail in the *SolidWorks Drawings* course.

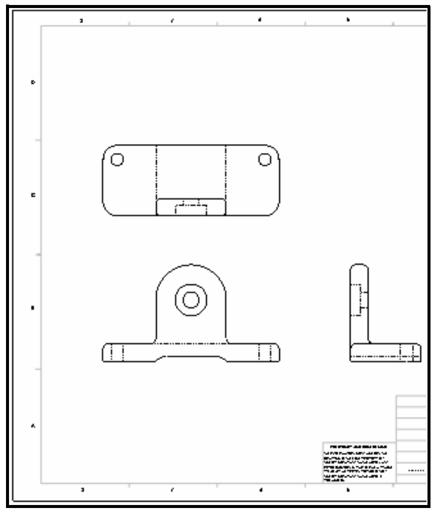
2 View Palette.
Clear **Import Annotations**. Drag the `Front` view from the **View Palette** and drop it onto the drawing as shown. It will be removed from the palette.



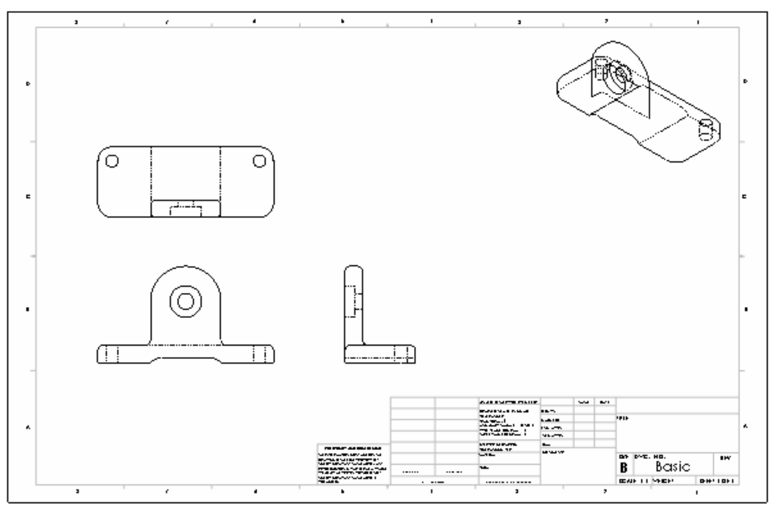
3 Projected views.
Add the Top view by moving the cursor above the view and clicking.

Return the cursor to the Front view and move to the right to create the Right view.

Click **OK**.

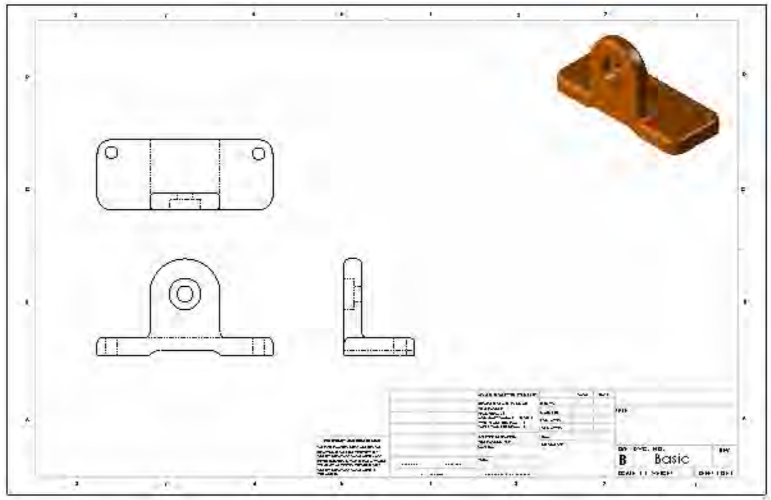


4 Drawing views.
Add the *Isometric view by dragging and dropping from the palette. Place it in the upper right corner.



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

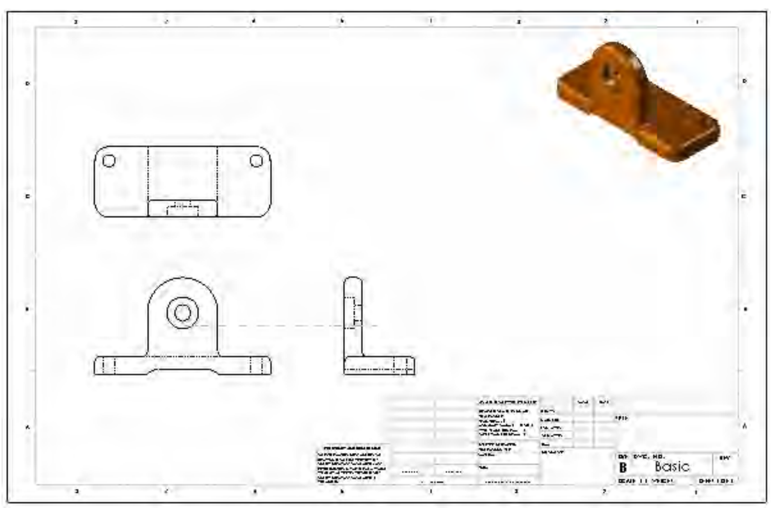
5 Display.
Click the Isometric view and click the **Shaded** button.



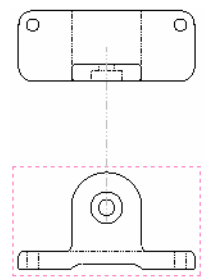
Moving Views

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.

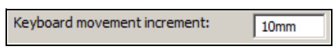


Tip Moving one of the projected views is limited by the alignment.



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

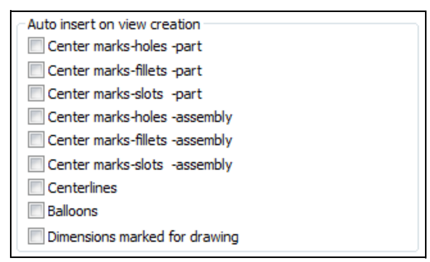
Tip Once the drawing view has been selected, it can be dragged with the mouse or moved with the arrow keys. The distance moved for each press of an arrow key is set under **Tools**, **Options**, **System Options**, **Drawings**, **Keyboard movement increment**.



Center Marks

Center Marks are attached to circle and arc centers in the drawing view.

Center marks were not 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.



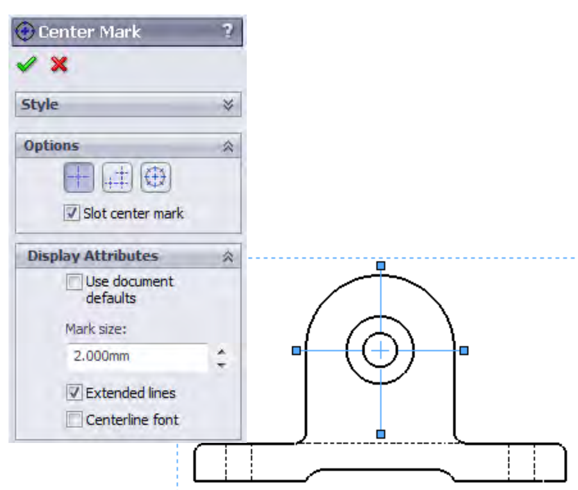
Where to Find It

- Click **Center Mark** on the Annotation toolbar.
- Or, click **Insert, Annotations, Center Mark**.

7 **Center Mark.**
Click the **Center Mark** icon and select the large arc in the front view. Clear **Use document defaults**, check the **Extended lines** option and set the **Mark size** to **2mm** as shown.

Repeat for selection of the two holes in the Top view.

Click **OK**.



Dimensioning

Dimensions can be created in drawing views using several tools. These dimensions are not related to the dimensions generated in model sketches and features. They are:

- **Smart dimensioning** - Uses the standard **Smart Dimension** tool to manually add dimensions like those in a sketch.
- **DimXpert** - Automates dimensioning by working from a datum position.

Driven Dimensions

These dimensions are considered to be *driven* dimensions. Driven dimensions always display the proper values but *cannot* be used to change the model.

Note

By default, dimensions of this type are displayed differently:

- They are displayed in a different color.
- The value is enclosed in parentheses (smart dimensioning).

Introducing: Dimension Assist Tool DimXpert

The **DimXpert Dimension Assist Tool** assists in dimensioning a view based on the selection of a datum and model geometry. Options for polar, linear, baseline, and chain styles are available.

Where to Find It

- Click **Smart Dimension** and **DimXpert**.

---

8 Datum setup.

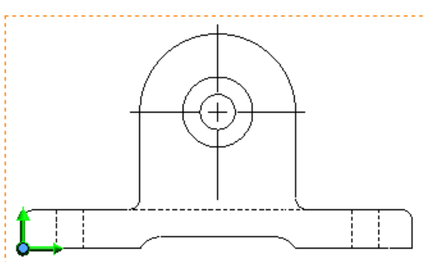
Click **Smart Dimension** and the **DimXpert** option.

Select **Linear dimensioning** as the **Pattern Scheme**.

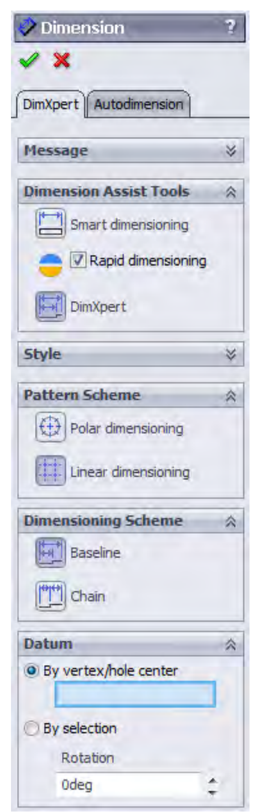
Select **Baseline** as the **Dimensioning Scheme**.

Select **By vertex/hole center** for the **Datum**.

For the **Datum**, click the lower left vertex in the Front view as shown.

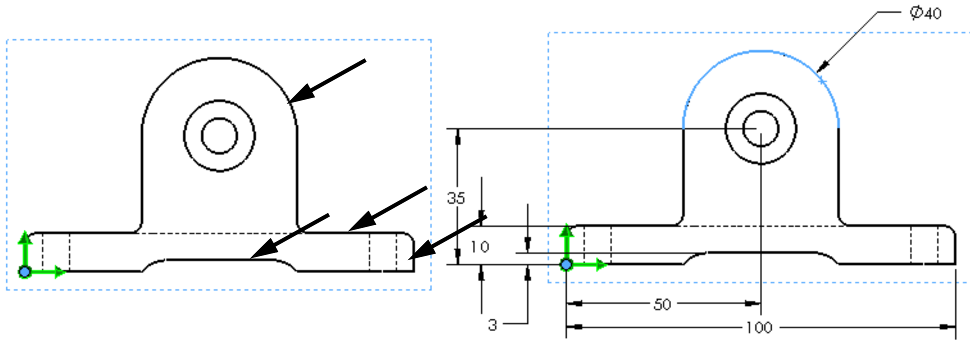


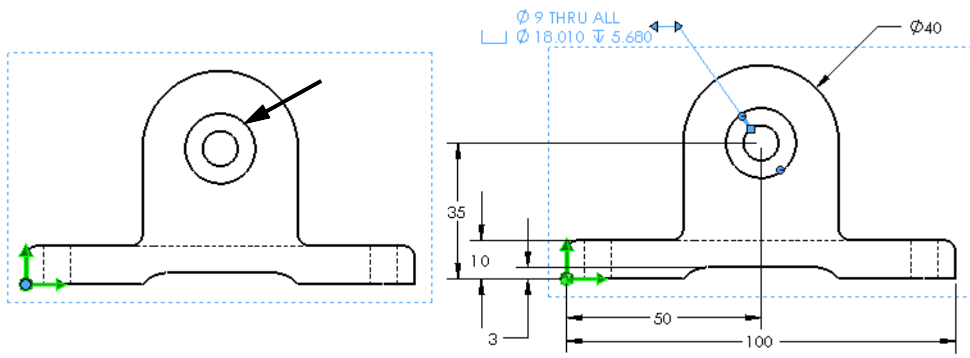
The datum selection can also be a hole center.



9 Edge selections.
Select the edges, linear and circular, as shown. The dimensions are added based on the position in reference to the datum.

10 Hole selection.
Select an edge of the hole feature as shown. The geometry is read as a counterbore and an appropriate dimension is added.





Tip

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.

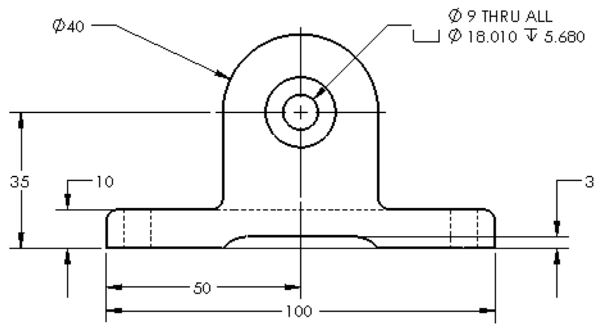
Click **OK** to complete the addition of dimensions to this view.

Manipulating Dimensions

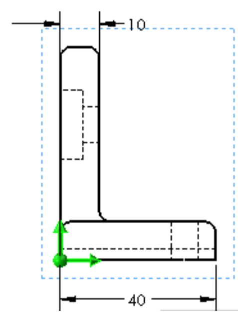
Once dimensions have been added to a view, there are several options as to how they can be manipulated:

Drag them into position.	Drag dimensions by their text to new locations. Use the inference lines to align and position them.
Hide them.	Right-click the dimension text and select **Hide** from the shortcut menu.
Move or Copy them to other views.	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.
Delete them.	Unwanted dimensions can be deleted from the drawing using the **Delete** key.

11 Repositioning dimensions.
Drag dimensions within the view to reposition them as shown.



12 Add to another view.
Using the same procedure, position the datum as shown and generate these dimensions.



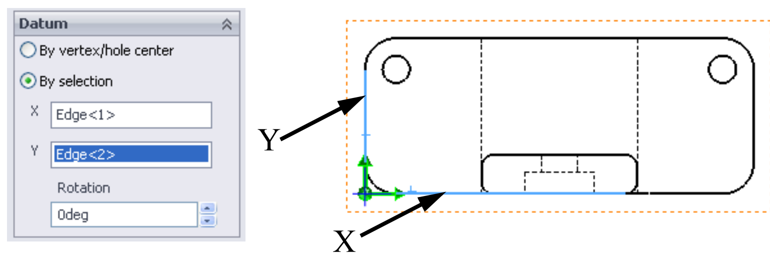
Datum by selection

The **Datum by selection** option is used where a virtual sharp edge is required as the datum. Two edges and an optional angle can be used to define it.

13 Datum by selection.

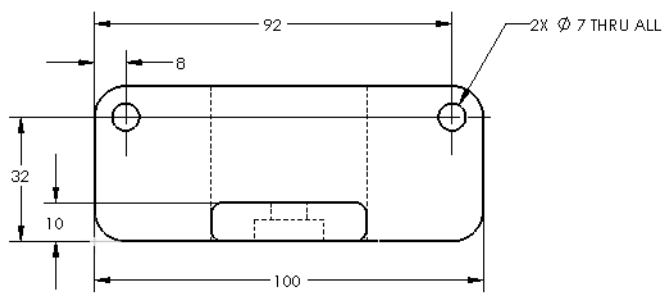
Click **Smart Dimension** and the **DimXpert** option. Select **Linear dimensioning** and **Baseline** as in the previous procedure.

Select **By selection** for the **Datum**. For the **Datum**, click the lower line in the Top view as **X** and the left vertical line as **Y**.



14 Dimension.

Select edges to create the dimensions as shown. Click **OK**.



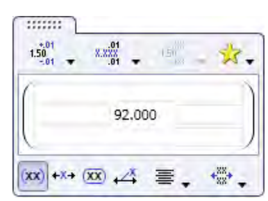
Note

Features of the same type and size are collected into single dimensions as seen for the hole and fillet dimensions.

15 Palette.

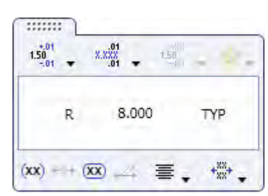
Click the **92mm** dimension and move the cursor onto the dimension palette icon that appears.

In the dialog, click **Add Parenthesis**. Repeat for the **32mm** and **8mm** dimensions that locate the holes.



Tip

The blank areas left, right, above and below the numeric text can be used to add text relative the numeric text. A common use would be to append a dimension with TYP as shown.



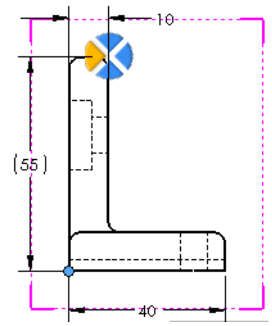
Dimension Assist Tool - Smart Dimensioning

Use the **Smart dimensioning** option of the dimension assist tool to manually add dimensions not created by the DimXpert.

For more information, see *Introducing: Dimension Assist Tool DimXpert* on page 94.

16 Dimensioning.
Click **Smart Dimension** and the **Smart dimensioning** option. Select vertices at the top and bottom. Click the left (orange) hemisphere to place the dimension to the left of the view.

Click **OK**.



17 Display options.
Move to the palette and clear the **Add 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:

18 Switch windows.
Press **Ctrl+Tab** and place the cursor over the part file to switch back to the part document window.

Changing Parameters

SolidWorks 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**.
- Or, 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**.
- Or, use the keyboard shortcut **Ctrl+R**.

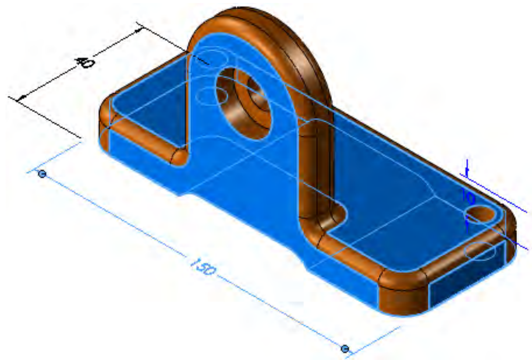
Rebuild vs. Redraw

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

19 Double-click on the feature.

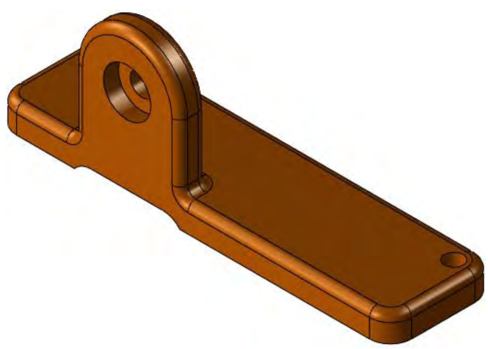
You can double-click on the `BasePlate` feature either in the FeatureManager design tree or the graphics area. When you do this, the parameters associated with the feature will appear.

Double-click on the **100mm** 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 **150mm**.



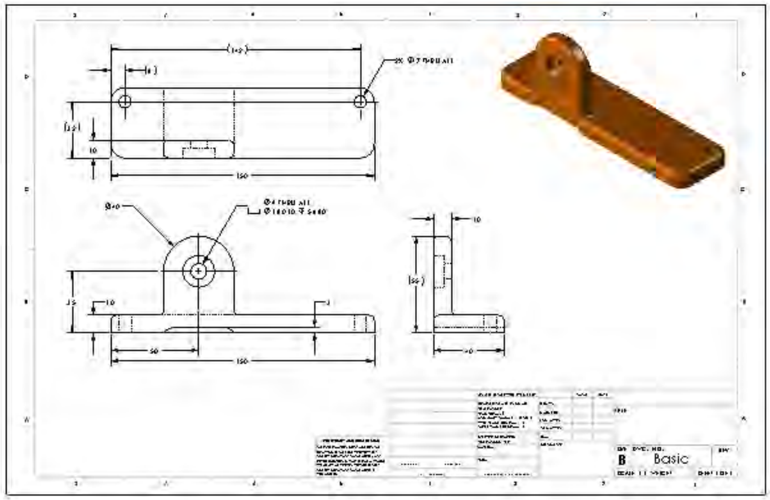
20 Rebuild the part to see the results.

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.



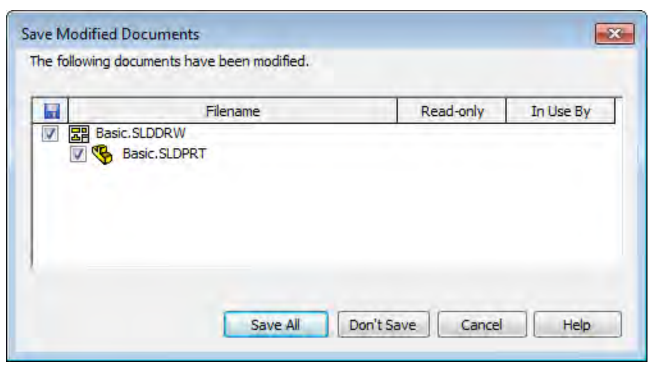
21 Update the drawing.

Switch back to the drawing sheet. The drawing will update automatically to reflect the changes in the model. Dimensions may move during the rebuilding process and require clean up.



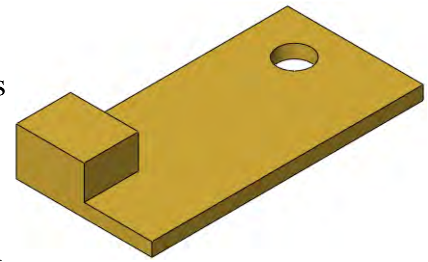
22 Close the drawing.

Click **File, Close** to close the drawing. Click **Save All** to save both the drawing and part files. Save the file in the same folder as the part.



Exercise 7: Plate

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



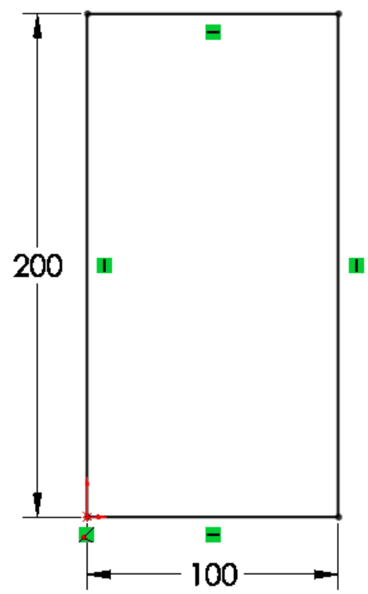
- *Choosing the Best Profile* on page 64.
- *Introducing: Corner Rectangle* on page 69.
- *Sketching on a Planar Face* on page 72.
- *Boss Feature* on page 71.
- *Using the Hole Wizard* on page 76.

Units: **millimeters**

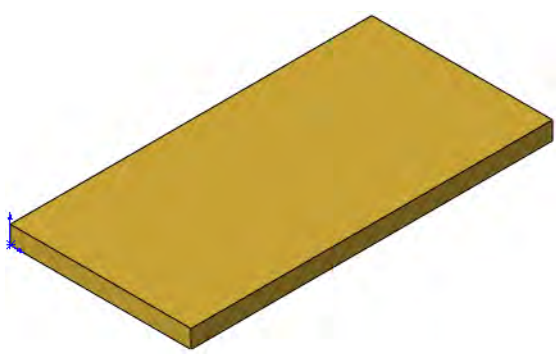
Procedure

Create a new mm part and name it `Plate`. Create the geometry as shown in the following steps.

1 **Sketch base feature.**
Create a new sketch on the Top plane. Add the geometry and dimensions as shown.



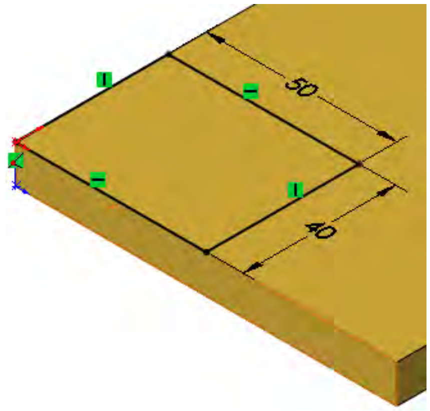
2 **Extrude base feature.**
Extrude to the sketch **5mm** as shown.

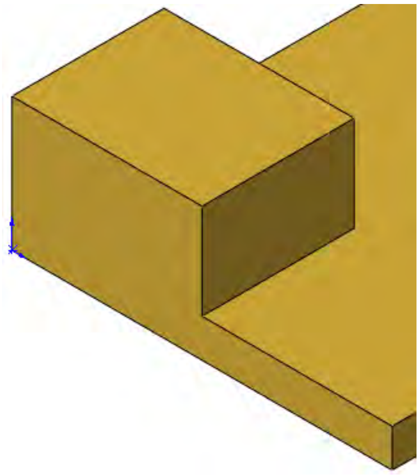


3 Boss.

Create a new sketch on the top face of the solid. Add the geometry and dimensions as shown.

Extrude a boss **25mm**.





4 Hole wizard.

Click the **Hole Wizard** and select the face shown.

Click the **Positions** tab. Place the points as shown.

Click the **Type** tab. Set the properties of the hole as follows:

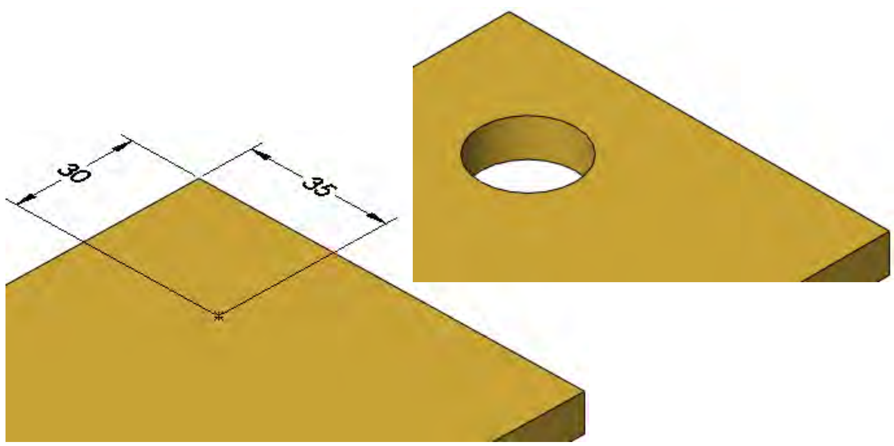
Type: Hole

Standard: Ansi Metric

Type: Drill sizes

Size: 25mm

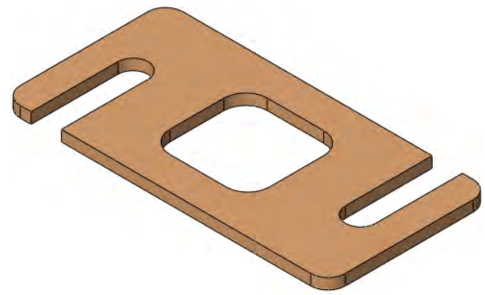
End Condition: Through All



5 Save and close the part.

Exercise 8: Cuts

Use rectangles, tangent arcs and cut features to create the part. This lab reinforces the following skills:



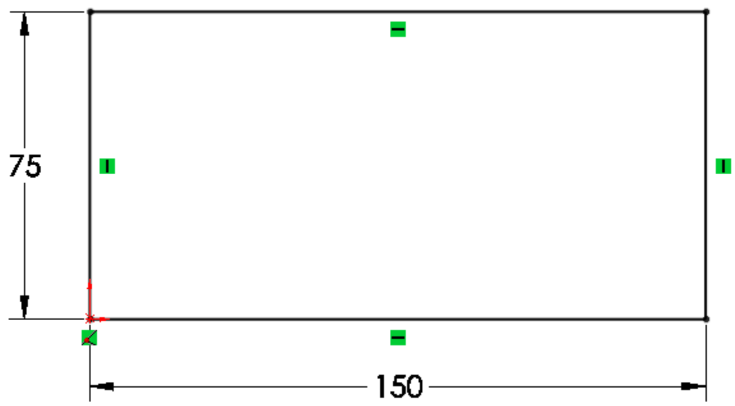
- *Introducing: Corner Rectangle* on page 69.
- *Tangent Arc Intent Zones* on page 72.
- *Cut Feature* on page 75.
- *Filleting* on page 78.

Units: **millimeters**

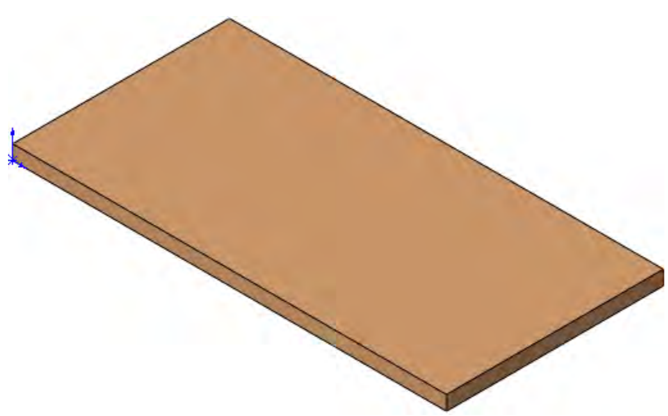
Procedure

Create a new mm part and name it Cuts. Create the geometry as shown in the following steps.

1 Sketch base feature.
Create a new sketch on the Top plane. Add the geometry and dimensions as shown.

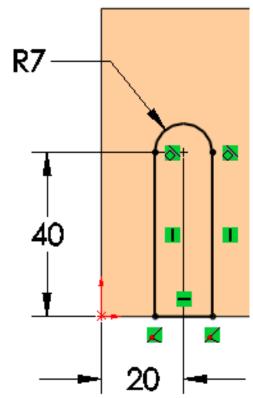


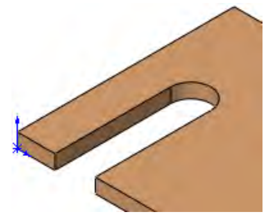
2 Extrude base feature.
Extrude to the sketch **5mm** as shown.



3 Cut slot.
Create a new sketch on the top face of the solid. Add the geometry and dimensions as shown.

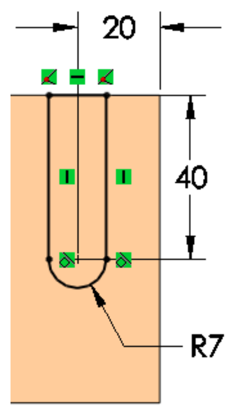
Extrude a cut using **Through All**.

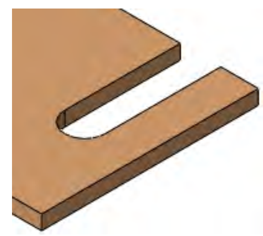




4 Cut another slot.
Create a new sketch using the same face. Add the geometry and dimensions as shown.

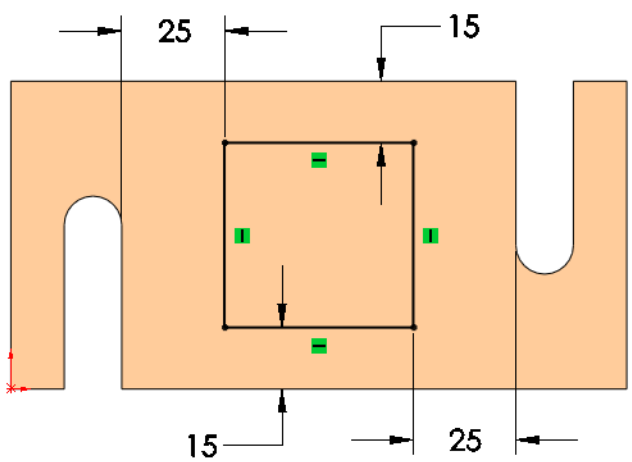
Extrude another cut using **Through All**.



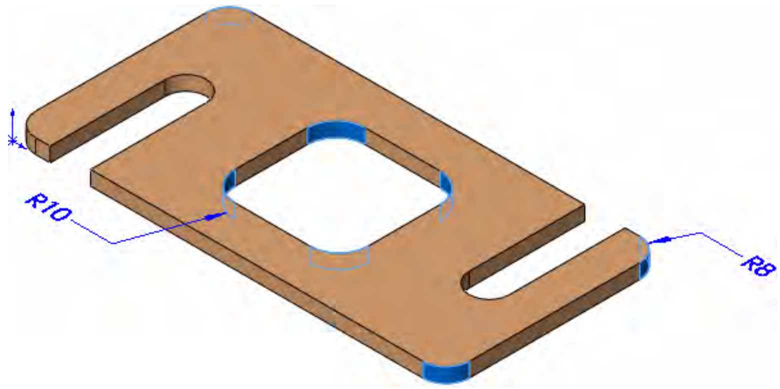


5 Cut rectangle.
Create a new sketch using the same face. Add the geometry and dimensions as shown.

Extrude another cut using **Through All**.



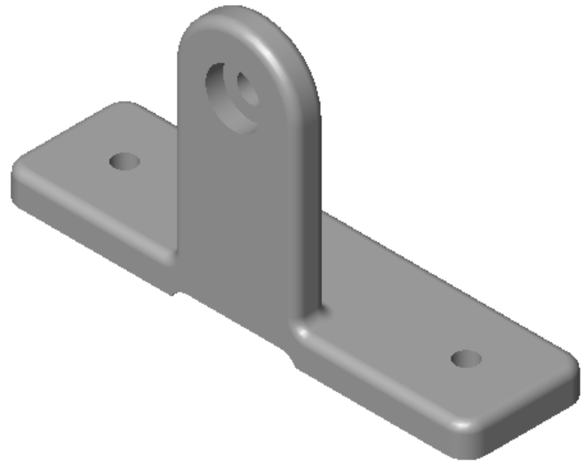
6 Fillets.
Add fillets of **R10mm** and **R8mm** to the edges as shown.



7 Save and close the part.

Exercise 9: Basic-Changes

Make changes to the part created in the previous lesson.



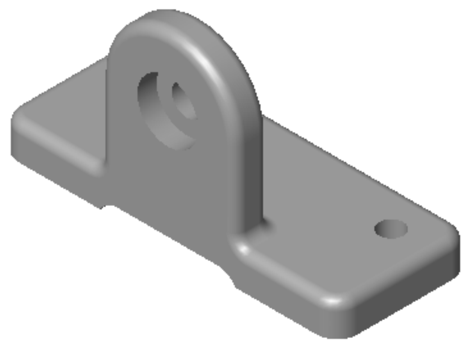
This exercise uses the following skills:

- *Changing Parameters* on page 99.
- *Rebuilding the Model* on page 99.

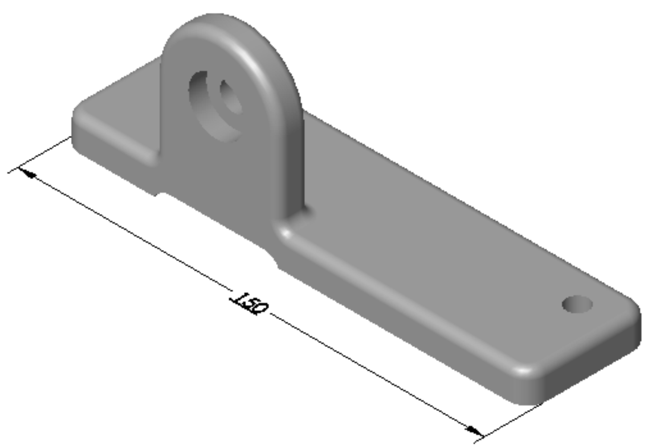
Procedure

Open an existing part in the `Lesson03\Exercises` folder.

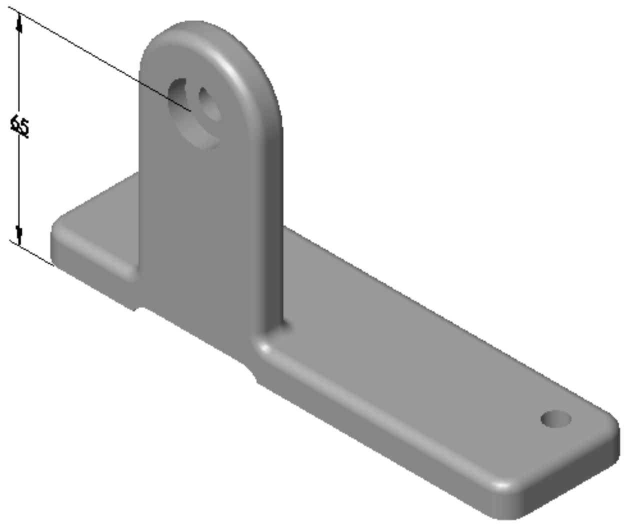
1 Open the part `Basic-Changes`.
Several changes will be performed on the model to resize it and check the design intent.



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 **150mm** (shown bold and underlined below) and rebuild the model.

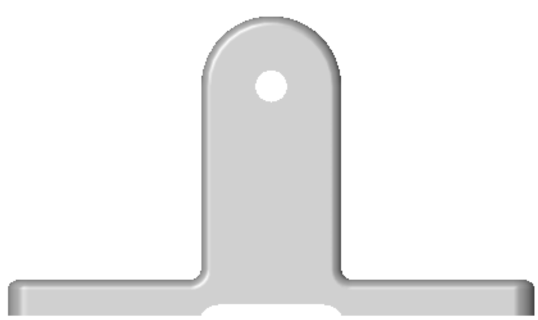


3 Boss.
Double-click the Vert boss feature and change the height dimension as shown. Rebuild the part.



4 Hole locations.
Double-click the 7mm hole feature and change the position dimensions to **20mm** each (only one is shown). Rebuild the model.

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



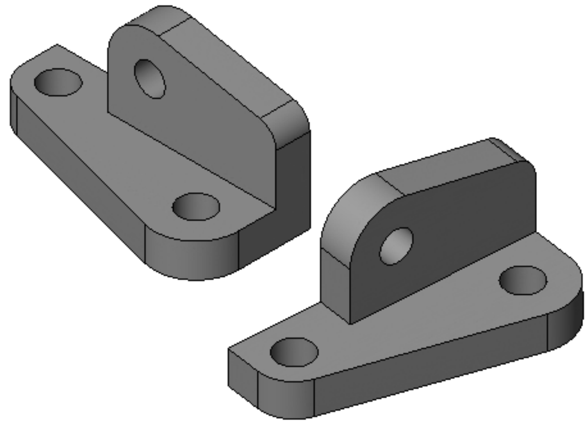
6 Save and close the part.

Exercise 10: Base Bracket

This lab reinforces the following skills:

- *Choosing the Best Profile* on page 64.
- *Boss Feature* on page 71.
- *Using the Hole Wizard* on page 76.
- *Filleting* on page 78.

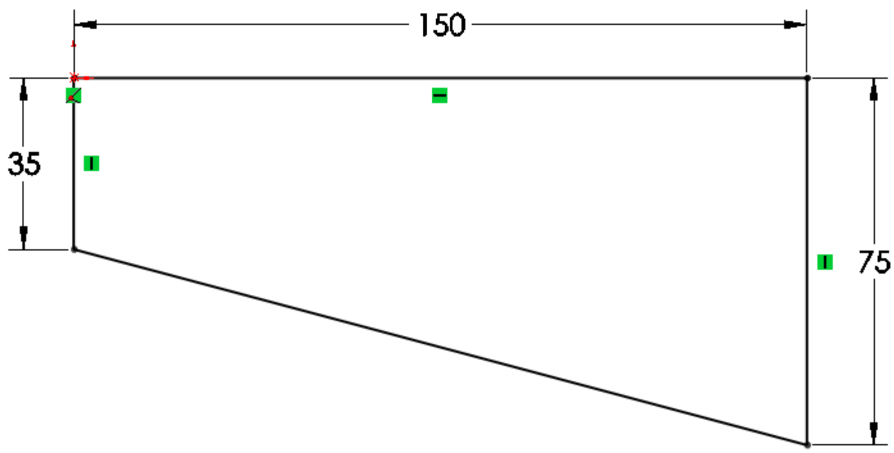
Units: **millimeters**



Procedure

Create a new mm part and name it Base_Bracket. Create the geometry as shown in the following steps.

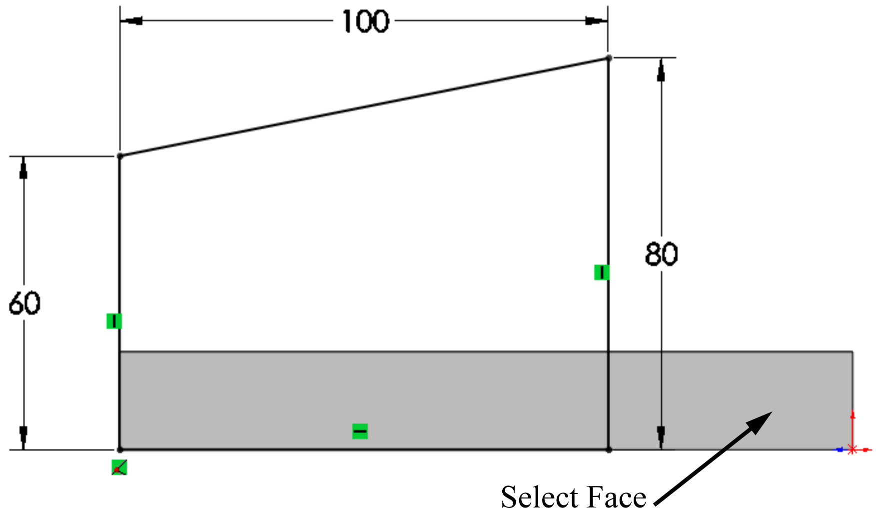
1 Sketch base feature.
Create a new sketch on the Top plane. Add the geometry and dimensions as shown.



2 Extrude base feature.
Extrude the sketch **20mm** to create the base feature as shown.

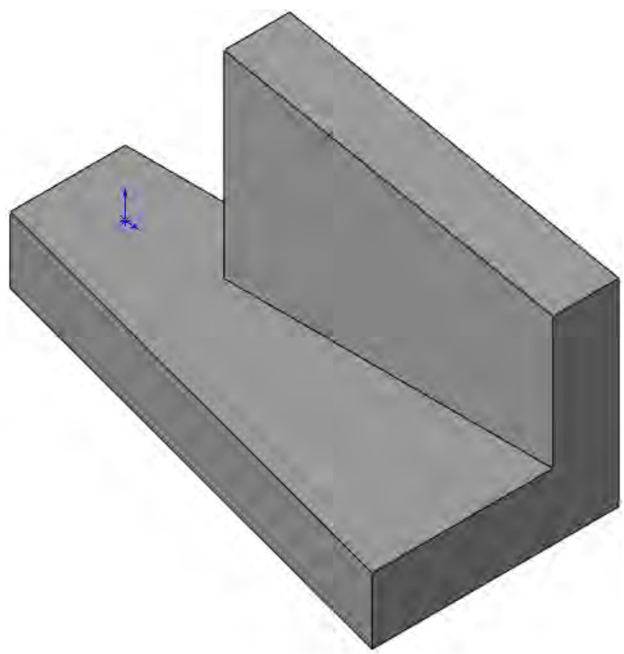
3 Sketch on rear face.

Change to the `Rear` view orientation, select the face indicated and create a new sketch. Add the geometry and dimensions as shown.



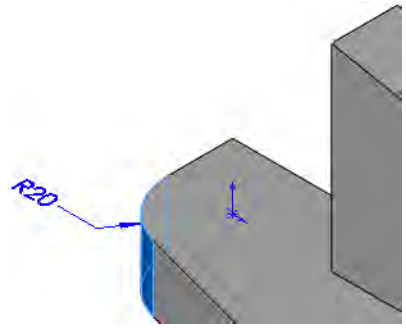
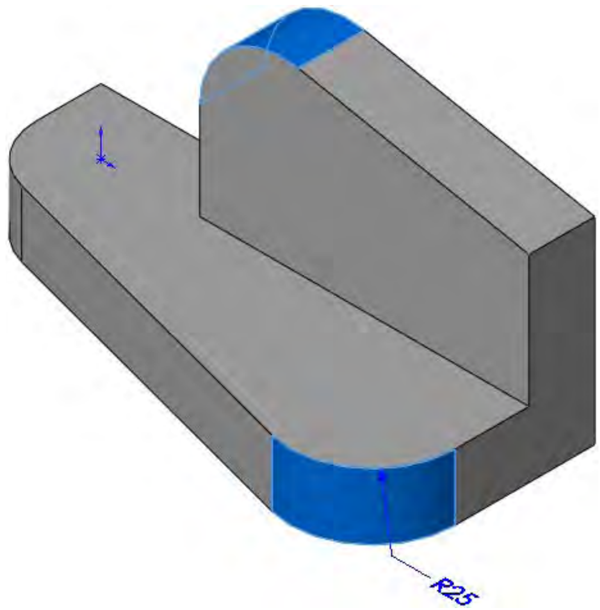
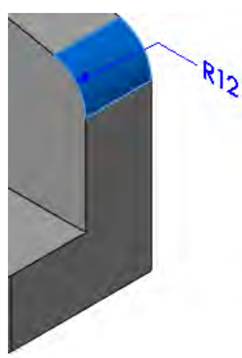
4 Extrude boss.

Extrude to the sketch **20mm** as shown.



5 Fillets.

Add fillets to the edges as shown.

R20mm	
R25mm	
R12mm	

6 Hole wizard.
Click the **Hole Wizard** and select the face shown.

Click the **Positions** tab. Use the centerpoints of the arcs to place the points as shown.

Click the **Type** tab. Set the properties of the hole as follows:

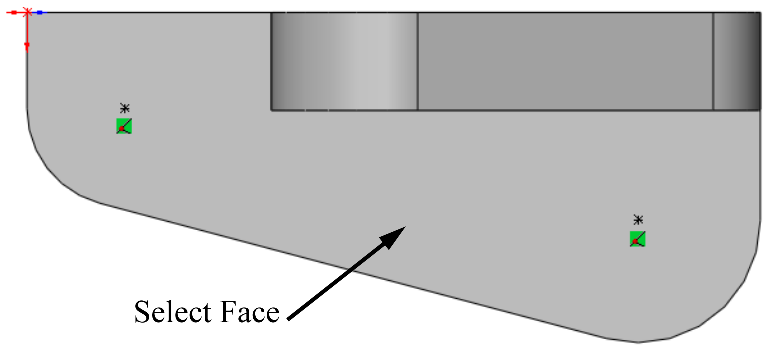
Type: Hole

Standard: Ansi Metric

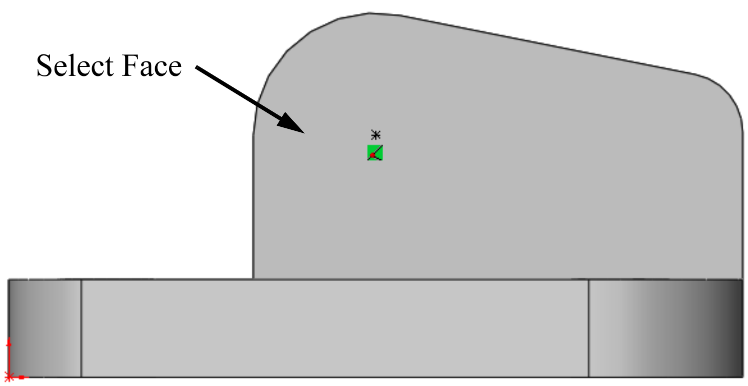
Type: Drill sizes

Size: 20mm

End Condition: Through All



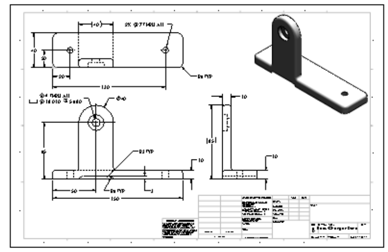
7 Second hole.
Repeat the procedure to create an **18mm** hole on a different face as shown.



8 Save and close the part.

Exercise 11: Part Drawings

Create this part drawing using the information provided.



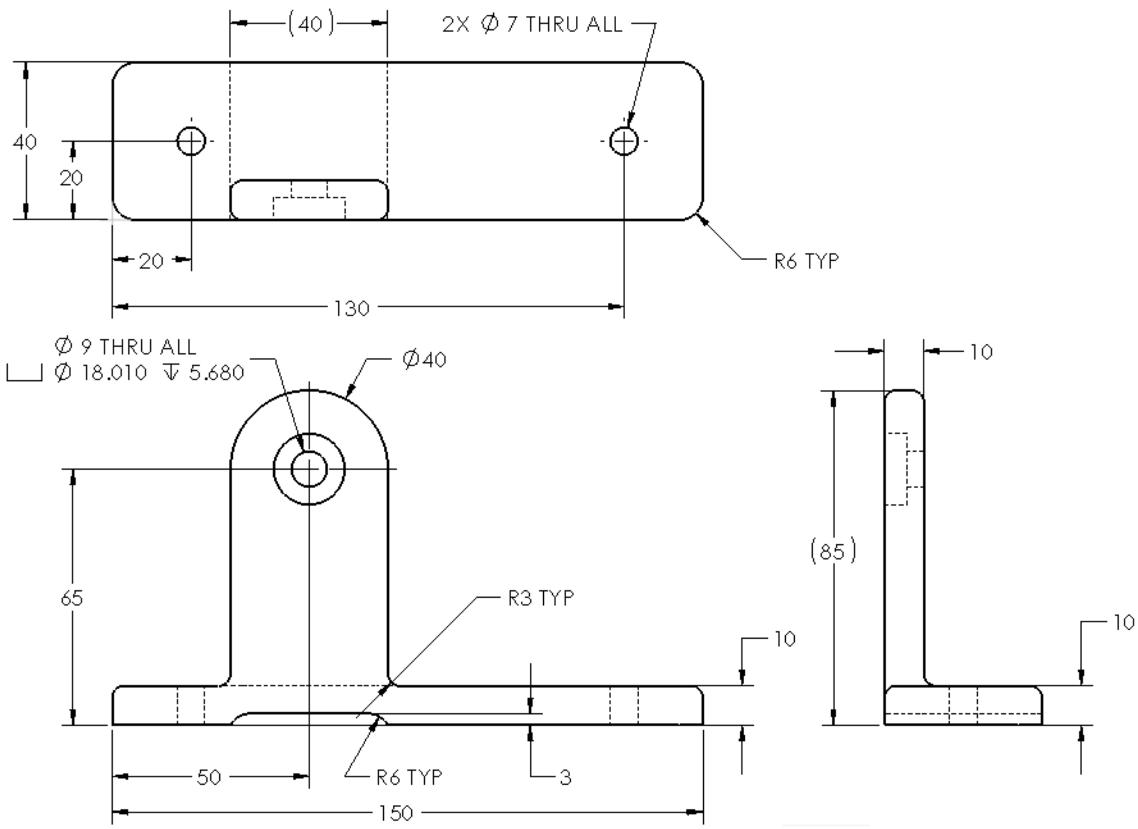
This lab reinforces the following skills:

- *New Drawing* on page 89.
- *Drawing Views* on page 90.
- *Center Marks* on page 93.
- *Dimensioning* on page 94.

Use the B_Size_ANSI_MM template and the built part Basic-Changes-Done.

Dimensioned View

Use the following graphics to create the drawing.



Lesson 4
Modeling a Casting or Forging

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

- Use the view display and modification commands.
- Copy and paste features.
- 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.

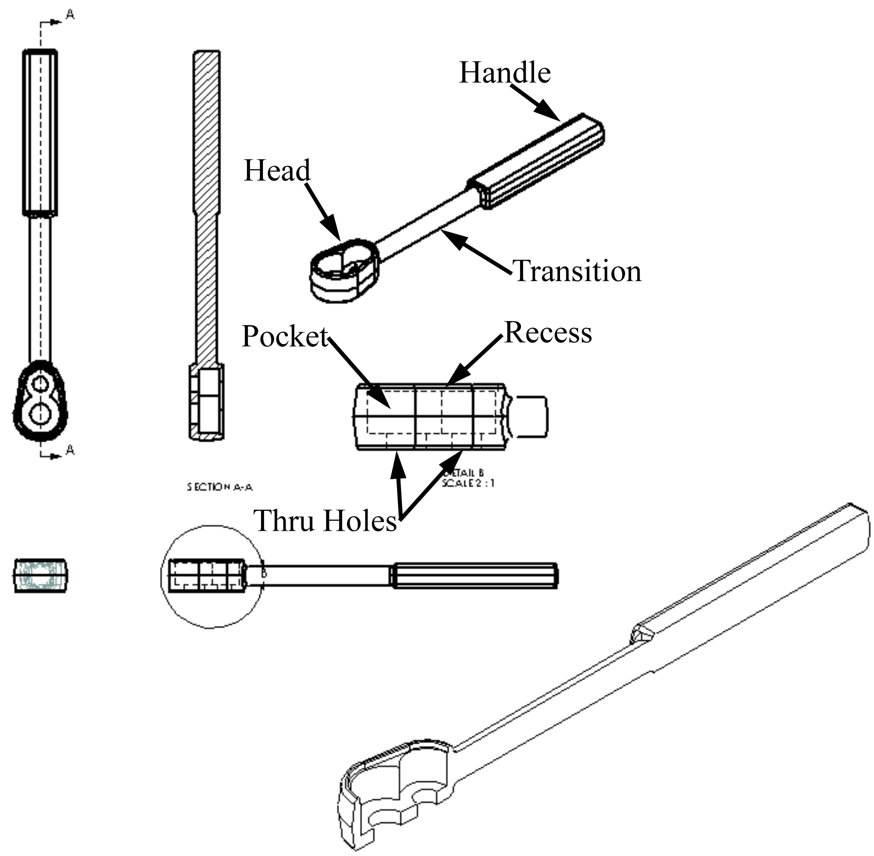
Stages in the Process

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.



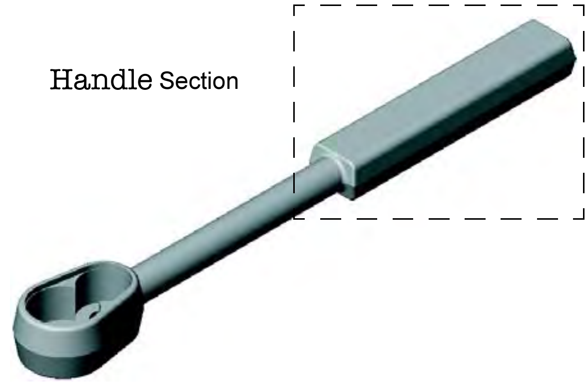
- **Centering:** The `Head`, `Handle` and `Transition` features are centered along an axis.
- **Symmetry:** The part is symmetrical, both with respect to a longitudinal centerline and with respect to the parting plane.

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.

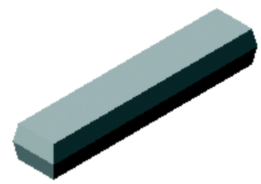
Building the Handle

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, a rectangular cross section.



- **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 **Sketch plane.**
Select the plane Top as the sketch plane.

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.

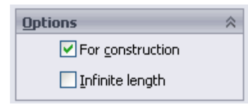
Where to Find It

- 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**.



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

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

Note

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

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.

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.

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

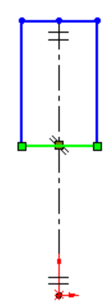
- From the **Tools** menu choose: **Sketch Tools, Mirror**.
- Or, on the Sketch toolbar, click **Mirror Entities**.

---

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.

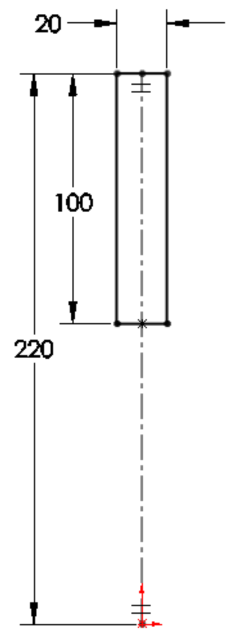
7 **Complete the sketch.**
Add a line in the vertical direction and then horizontal, stopping at the centerline. Turn off the mirror tool.



Tip

Do not cross the centerline while sketching in the **Dynamic 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.

8 Dimensions.
Fully dimension the sketch.



Mid Plane Extrusion

In this part, the first feature created is a **Mid Plane** extrusion. The Mid Plane option extrudes the profile equally in opposite directions. The depth is the total extruded distance, split evenly in each direction.

Draft Toggle

The **Draft** toggle can be selected to draft the faces normal to the extrusion direction. The **Draft angle** and **Draft outward** toggles can be used to set the value and direction of the angle.

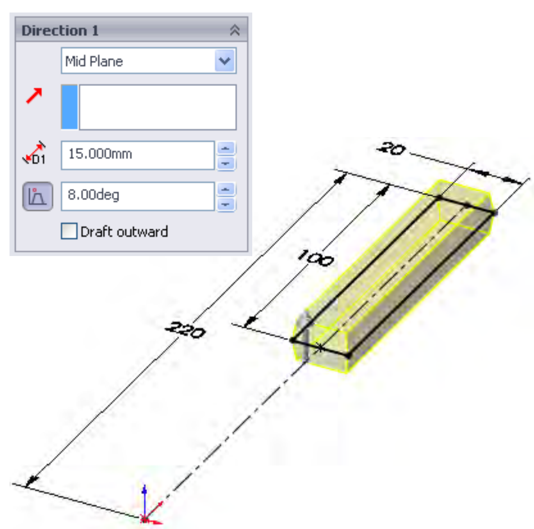
9 Base/Boss Extrusion.

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

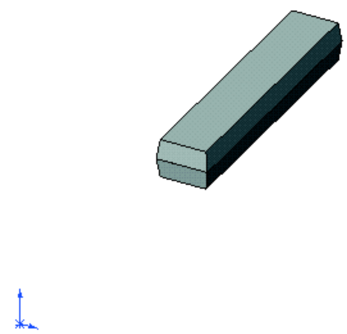
10 Extrusion.
Choose the **Mid Plane** option from the list and enter a depth of **15mm**.

Click **Draft** and set the angle to **8°**. The **Draft outward** check box should be cleared.

Click **OK** to create the feature.

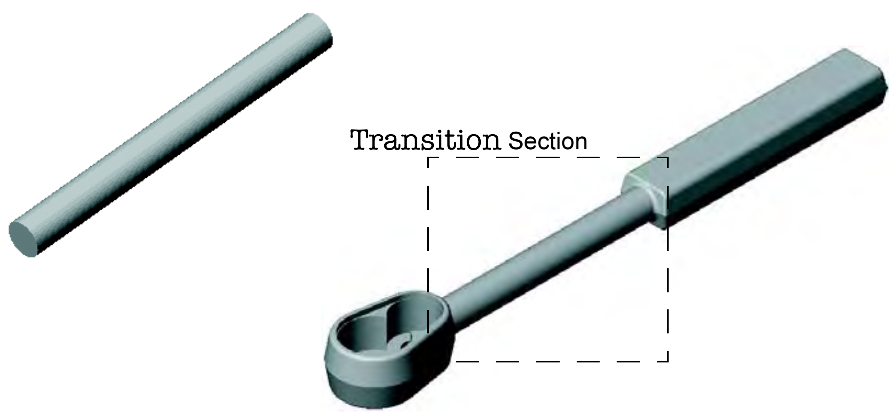


11 Completed feature.
The completed feature is shown at the right.
Name the feature Handle.



Sketching Inside the Model

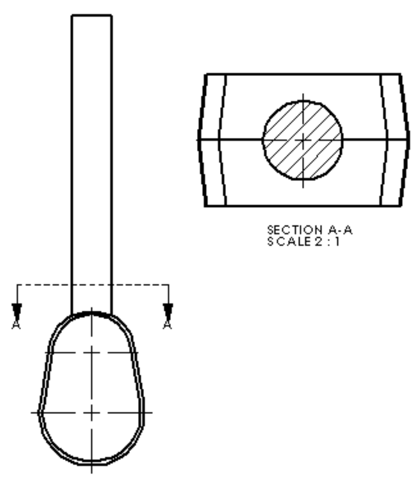
The second feature in the part is the Transition, another boss that will connect to the Handle feature. The sketch for this feature is created on a standard plane.



Design Intent of the Transition

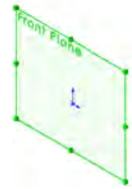
The Transition feature is a simple circular profile that is extruded up to the existing Handle feature.

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



12 Showing the Front plane.

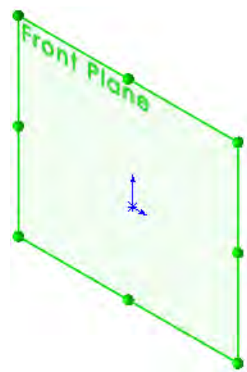
Switch to an Isometric view and select the Front plane from the FeatureManager design tree. It will be highlighted on the screen.



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

13 Plane settings and changes.

There are settings to determine how 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.

14 Open a new sketch.

With the Front plane still selected, click the **Sketch** tool.

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 plane, sketch, planar face or feature that contains a sketch.

Tip

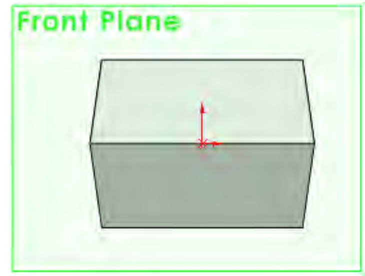
Clicking the **Normal To** icon a second time will flip the orientation around to the opposite side of the plane.

Where to Find It

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

15 Normal To view orientation.
Using the **View Orientation** dialog box, change to the **Normal To** orientation by double-clicking **Normal To**.

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

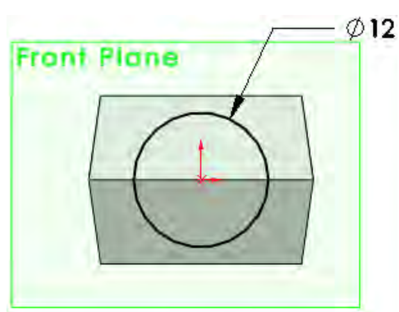
- From the **Tools** menu, select **Sketch Entities, Circle** or **Perimeter Circle**.
- Or, on the Sketch toolbar, click **Circle** or **Perimeter Circle**.

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.

16 Add a circle and dimension it.
Using the **Circle** tool, add the circle at the origin.

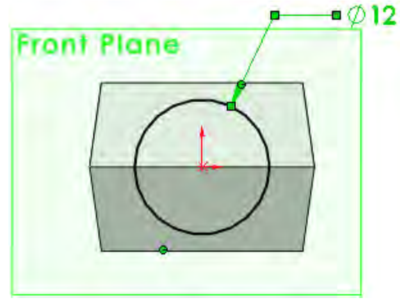
Add the diameter dimension to fully define the sketch. Set the value to be **12mm**. The sketch is fully defined.



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.

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

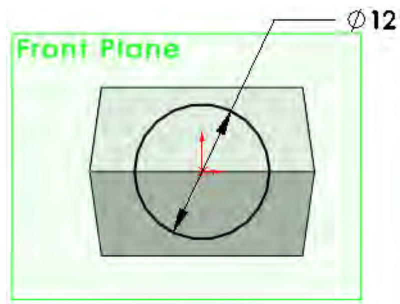


Note

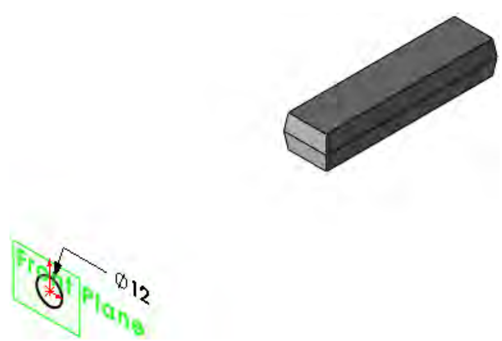
The highlighted geometry can appear in any color. The color depends on the **Selected Item 1** color setting.

18 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.



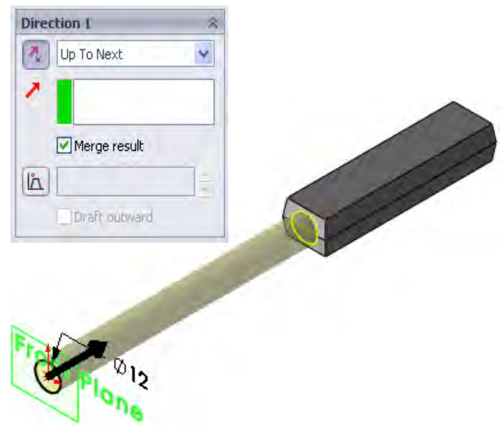
19 Change to Isometric view.
Unlike when you created the first feature, the system will not switch view orientations automatically for any other bosses or cuts. Change to an Isometric view.



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.

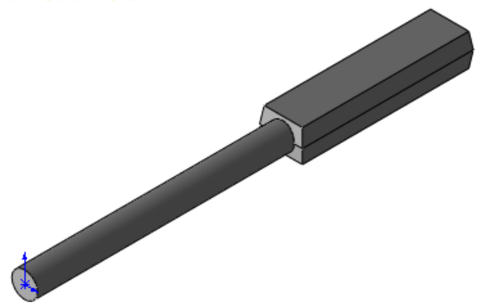
20 Up To Next extrusion.
Click **Insert, Boss/Base, Extrude...** and watch the preview display. Change the direction so that the preview shows the extrusion running towards the Handle.



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

Click **OK**.

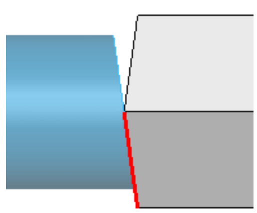
Rename the feature to Transition.

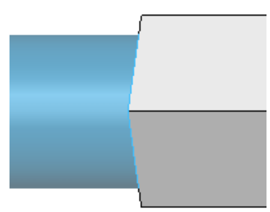


21 Hide plane.
Right-click the Front plane and click **Hide**.

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.



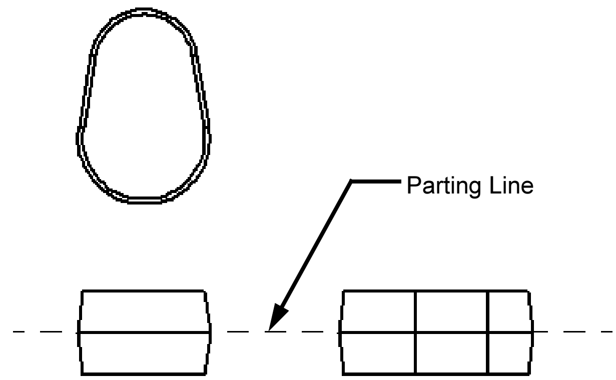


Design Intent of the Head

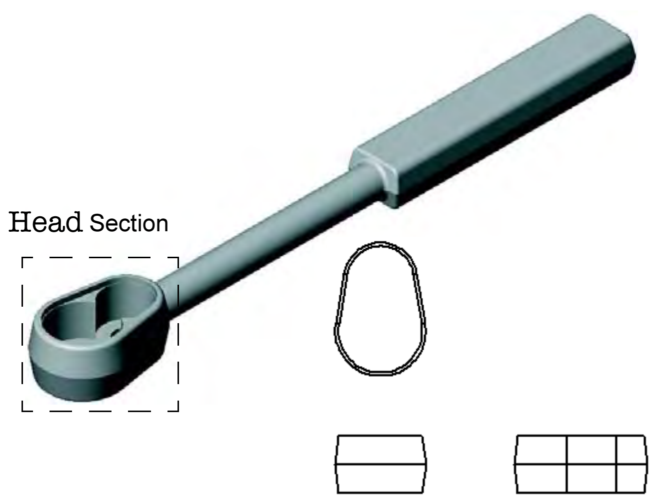
The Head is a sketched feature that uses lines and tangent arcs to form the basic outline or profile. The profile is extruded in opposite directions, equally, with draft. This feature is the key feature of the part. It will contain pockets and holes used for the location of other parts.

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.

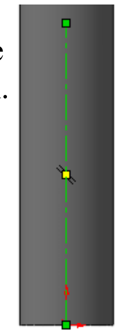


- **Profile Location:** The sketch geometry is located on the parting plane of the solid with the larger arc centered with respect to the model origin.
- **Draft:** The applied draft is equal on both sides of the parting plane.
- **Thickness:** The thickness of the part is equal on both sides of the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



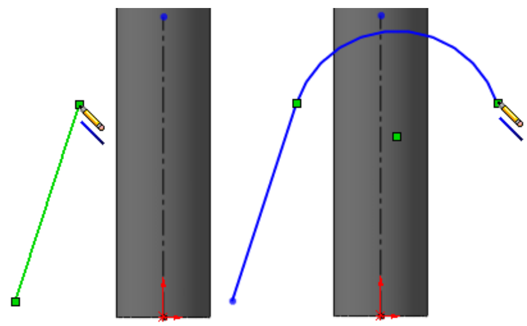
22 Centerline.

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



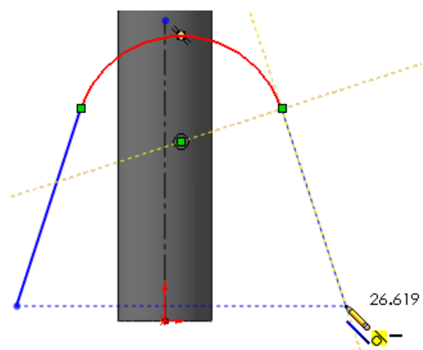
23 Lines and arcs.

Create a line and transition directly into a tangent arc using **Autotransitioning**. For more information, see *Autotransitioning Between Lines and Arcs* on page 73.



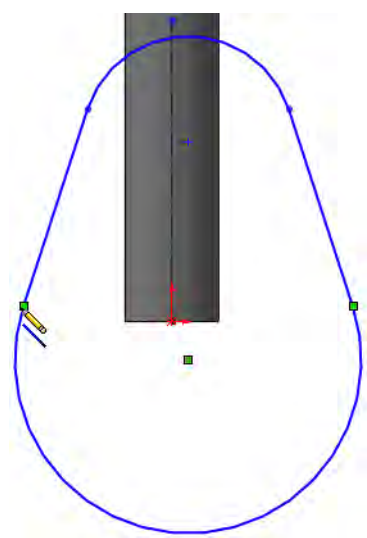
24 Return to a line.

After the arc is complete, the line tool again becomes active. Sketch a line using the tangent inference line. End the line at the end of the first line.

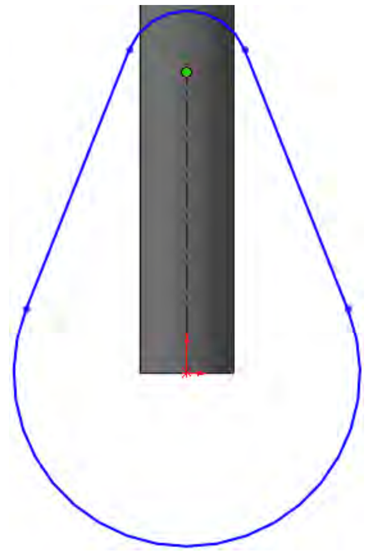


25 Tangent arc.

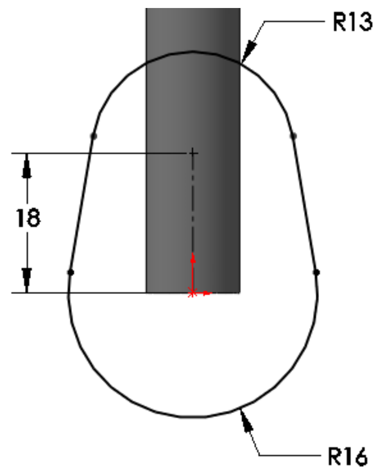
Use autotransitioning again to close the profile starting with a tangent arc from the last endpoint.



26 Add relations.
Add relations to attach the arc centerpoints to the ends of the centerline as shown.



27 Dimensions.
Add dimensions as shown to fully define the sketch.



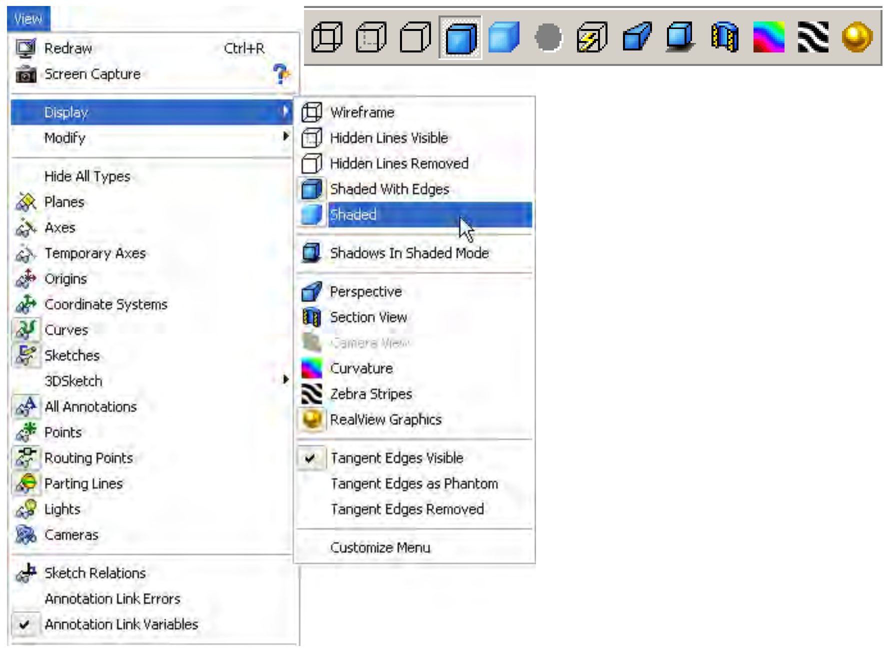
28 The extrusion.
Change to an Isometric view and click **Insert, Boss/Base, Extrude...** from the menu. Set the type to **Mid Plane**, depth to **20mm** and draft to **8°**. Rename the latest feature to Head.

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

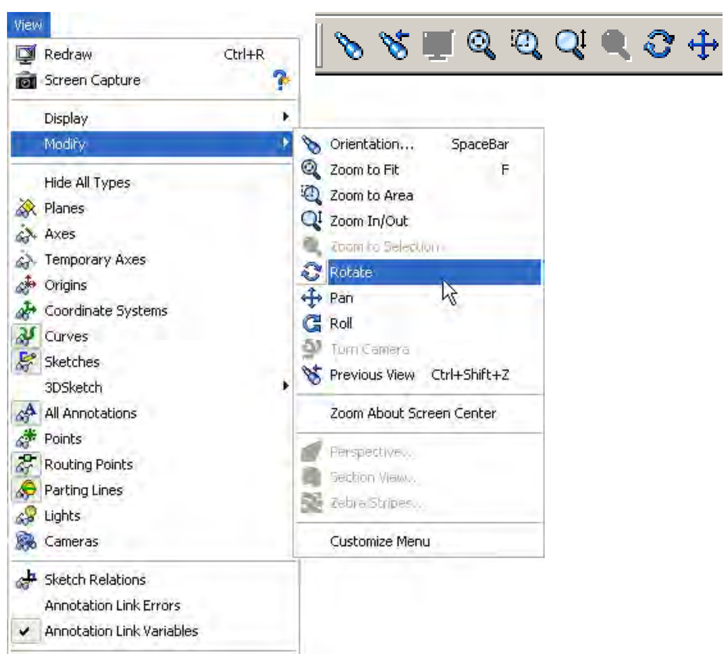
View Options

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.

- **Display Options**



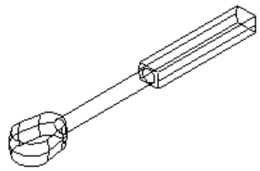
- **Modify Options**

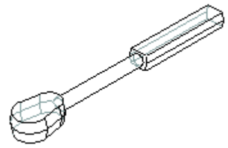


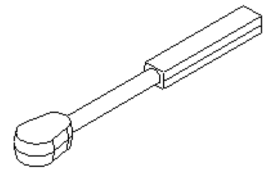
Note The lower portions of these menus have been truncated.

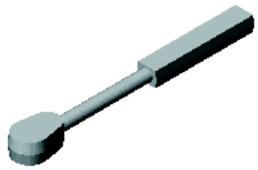
Display Options

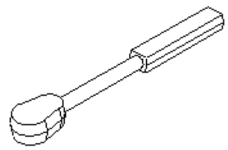
The following illustrations of the `Ratchet` illustrate the different types of display options.

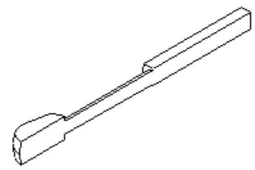

Wireframe


Hidden Lines Visible


Hidden Lines Removed

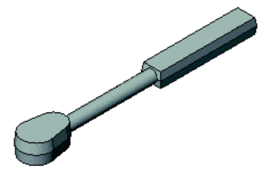

Shaded


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.

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.
	**Roll View:** Rotates the view about an axis normal to the screen 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 or wheel mouse button on a three button mouse can be used to dynamically manipulate the display. Using the middle or wheel mouse button you can:

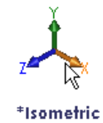
Function	Button	Wheel
Rotate	Press and hold the middle mouse button. As you move the mouse, the view rotates freely.	Press and hold the wheel mouse button. As you move the mouse, the view rotates freely.
Rotate about geometry	Click the middle mouse button on the geometry. As you move the mouse, the view rotates about that selected geometry. The geometry can be a vertex, edge, axis, or temporary axis.	Click the wheel mouse button on the geometry. As you move the mouse, the view rotates about that selected geometry.
Pan or Scroll	Press and hold the **Ctrl** key together with the middle mouse button. The view will scroll as you drag the mouse.	Press and hold the **Ctrl** key together with the wheel mouse button. The view will scroll as you drag the mouse
Zoom	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.	Spin the wheel mouse button. The view will zoom larger as you spin the wheel downward; smaller as you spin the wheel upward.
Zoom to Selection	Double-click the middle mouse button to zoom the entire model to fit.	Double-click the middle mouse button to zoom the entire model to fit.

Tip

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

Reference Triad Functions

The **Reference Triad** can be used to change the view orientation. Selecting an axis, with or without additional keys, can be used to control rotations.



Selection	Result
Select axis not normal to screen	Axis direction is normal to the screen.
Select axis normal to screen	Rotate view 180° clockwise.
Shift + select axis	Rotate view 180° counterclockwise.
Alt + select axis	Rotate view using **Arrow key** increments.

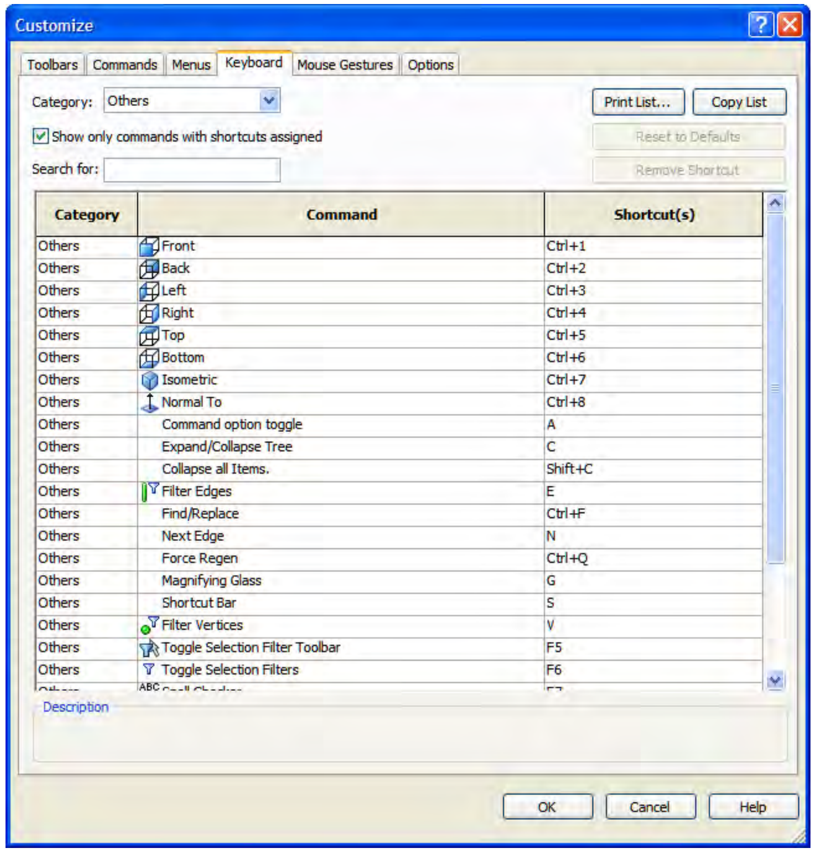
Keyboard Shortcuts

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
g	Magnifying Glass
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

Tip Clear **Zoom to fit when changing to standard views** to prevent fitting the model to the graphics area on an orientation change.

Note Click **Tools, Customize** and the **Keyboard** tab to see assigned shortcuts. Use the same dialog to add shortcuts of your own.



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

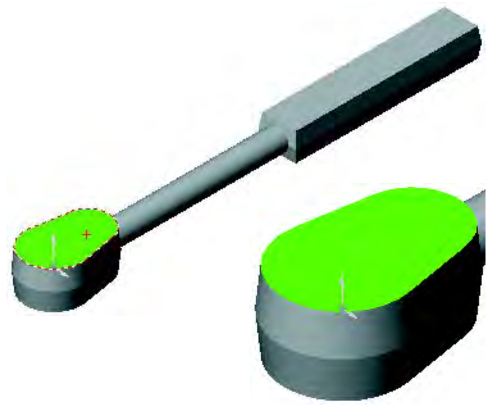
The **Zoom to Selection** option zooms in on a selected entity, making it fill the screen.

Where to Find It

- Select geometry and from the **View** menu, select **Modify, Zoom to Selection.**
- Or, right-click geometry and select **Zoom to Selection** .

Note Multiple geometry selections can also be used.

29 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.

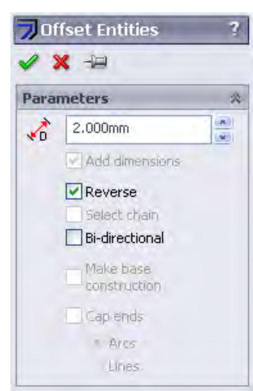
Introducing: Offset Entities

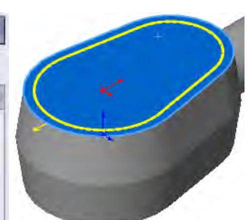
Offset Entities is used to create copies of model edges in a sketch. These copies are offset from the original by some specified amount.

Where to Find It

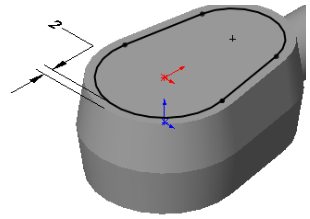
- From the **Tools** menu, select **Sketch Tools, Offset Entities...**.
- Or, on the Sketch toolbar, click **Offset Entities**.

30 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 and click **OK**.

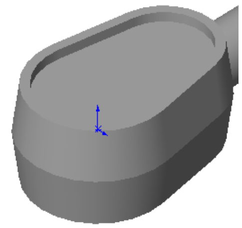




31 Resulting Offset.
The offset creates two lines and two arcs. This geometry is dependent on the solid face it came from and will change with the solid. The sketch is automatically fully defined and ready to extrude as a cut.



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

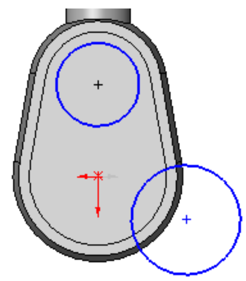


33 Rename the feature.
Change the name of the feature to Recess.

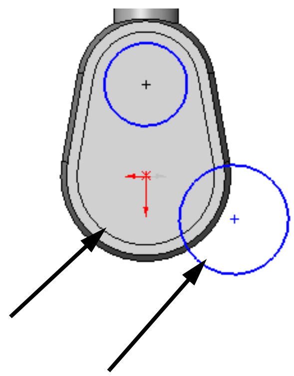
Creating Trimmed Sketch Geometry

The Pocket is another extruded cut feature. This sketch uses overlapping circles that are trimmed to create a single contour. The centers of the circles are related to existing circular centerpoints.

34 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.



35 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

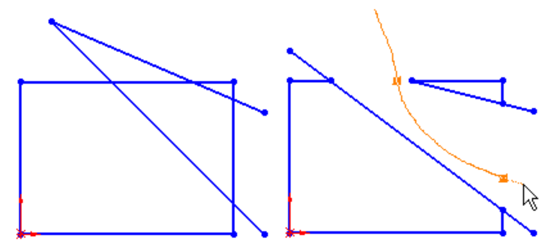
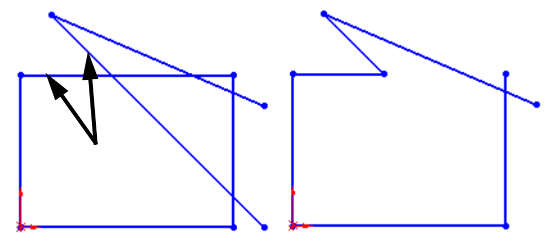
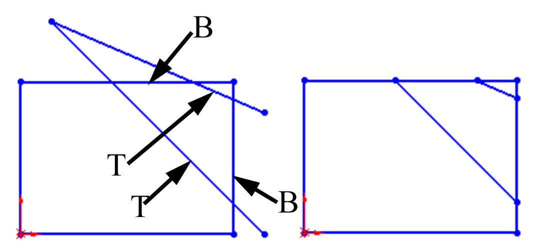
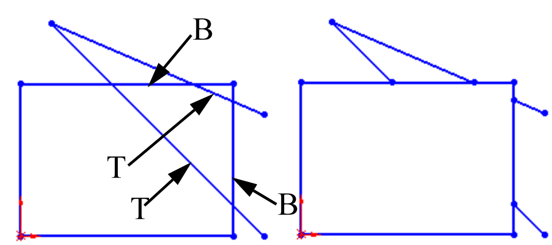
Sketch entities can be trimmed shorter using the **Trim** option. In this example, the overlapping portions of the circles will be removed. There are several trimming options: **Power Trim**, **Corner**, **Trim away inside**, **Trim away outside** and **Trim to closest**. They can also be lengthened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

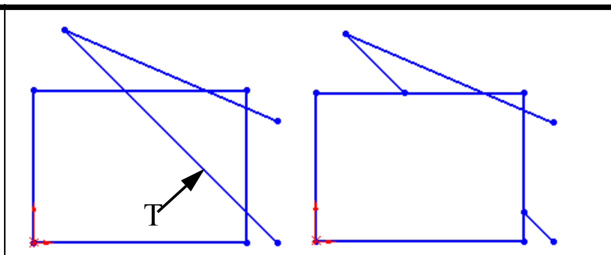
Introducing: Trim

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 of geometry is 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 of geometry against a boundary. Select the two boundaries (B) first then the pieces of geometry to trim (T).	

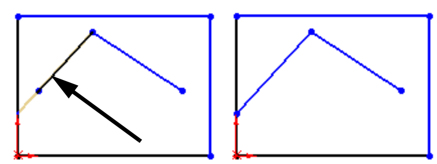
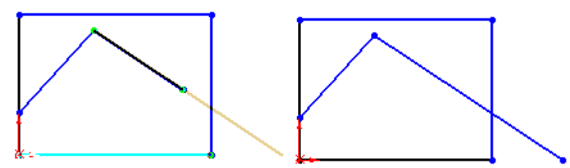
Use **Trim to closest** to trim selected geometry to the nearest intersection or remove a portion of the geometry between boundaries.	

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.	

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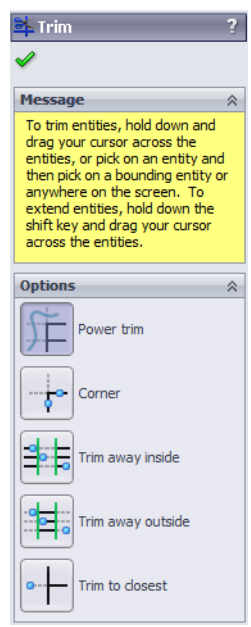
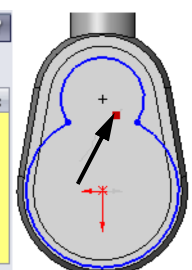
36 Trim the circles.

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



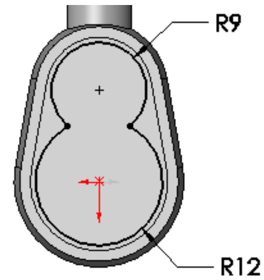


Rule

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.

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

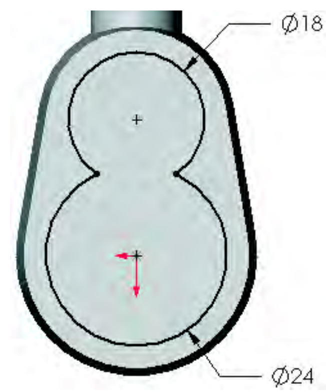


38 Turn off the dimension tool.
An easy way to turn off the dimension tool is to simply press the **Esc** key on the keyboard.

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**.

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

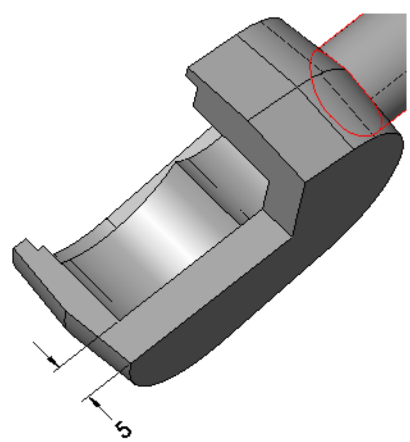


Introducing: Offset From Surface

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.

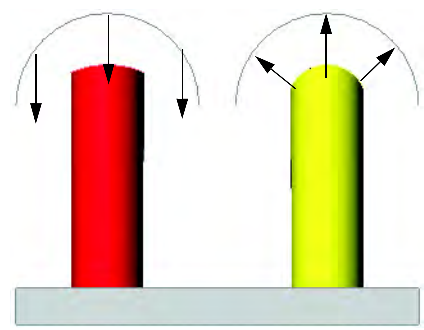
The **Translate Surface** option can be checked or cleared. Its meaning is explained below.



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.

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40 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.

Where to Find It

- Right-click on a face and select **Select Other**.

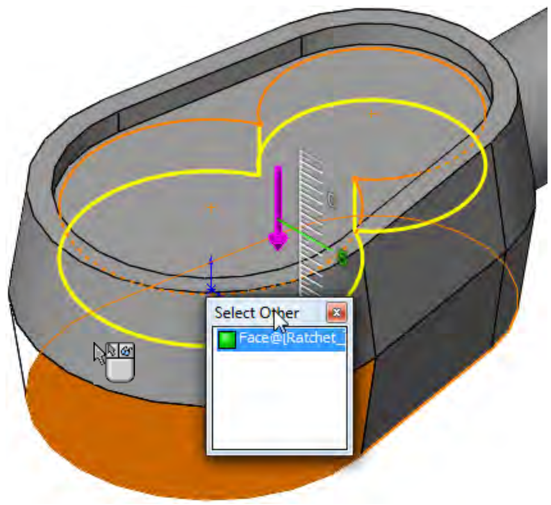
Select Other Procedure

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. Moving over faces listed 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.

41 Face selection.

Right-click over the hidden bottom face and choose **Select Other**. If there are multiple selections, 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 **Face** from the list.



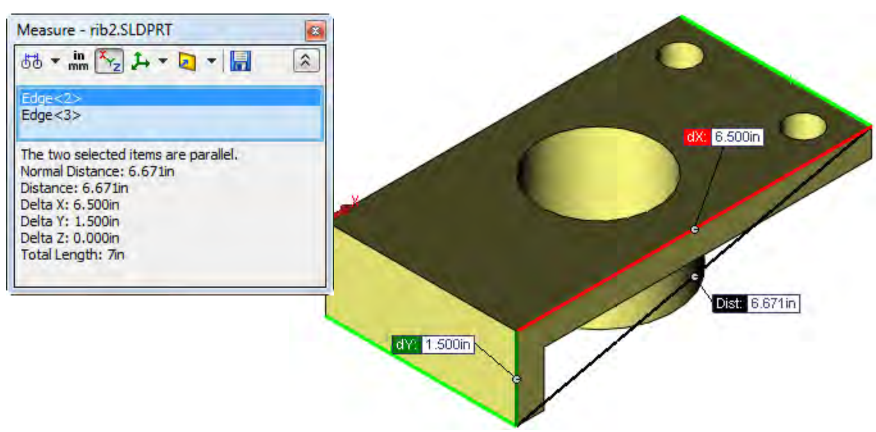
Rename the feature `Pocket`.

Tip

Other faces can be hidden during the selection. Right-click a face to hide it. Press **Shift** and right-click to unhide it and remove it from the list.

Measuring

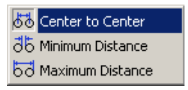
The **Measure** option can be used for many measurement tasks including measuring a single entity or between two entities. The measurement appears in the default units of the part.

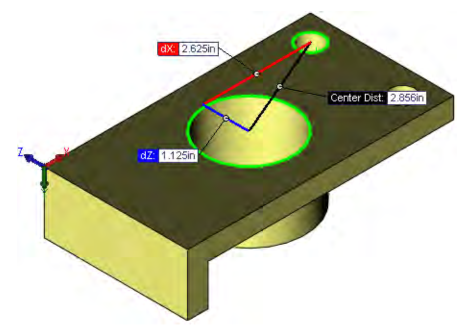


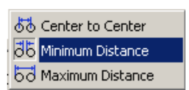
In this example it will be used to measure the shortest distance between an edge and a plane.

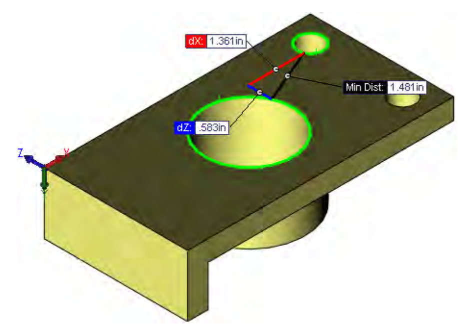
Introducing: Measure

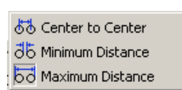
The **Measure** command can calculate distances, lengths, surface areas, angles, circles and X, Y, Z locations of selected vertices. For circles and arcs, the center, minimum and maximum dimensions are available as shown below.

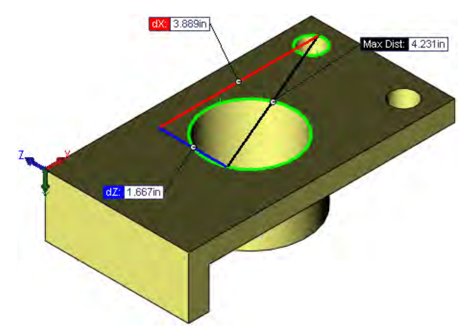


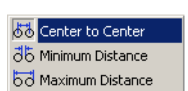


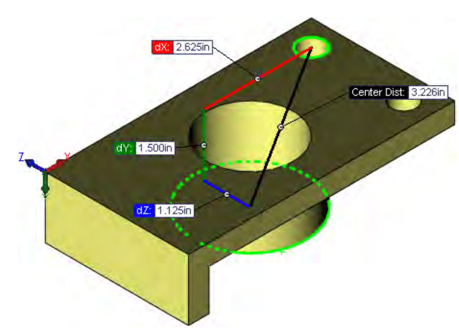










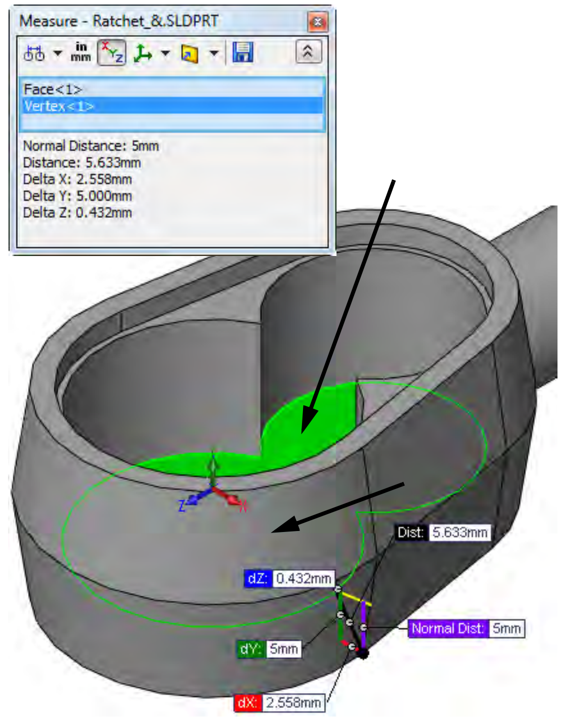


Where to Find It

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

42 Measure between face and vertex.
Click the **Measure** tool  and select the face (from step **41**) and vertex as shown.

The **Normal Distance** and **Delta Y** are both **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**.

Radius: 12mm Center: 0mm,8mm,0mm

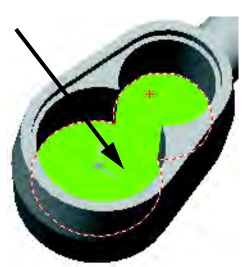
Using Copy and Paste

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.

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



44 Create a circular hole.
Sketch a circle concentric to the upper circular edge as shown. Add a dimension, set the diameter to **9mm** and create a **Through All** cut.
Name the feature Wheel Hole.

Copy and Paste Features

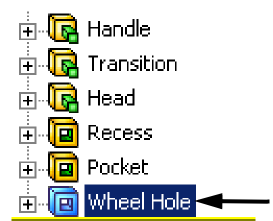
Simple sketched features and some applied features can be copied and then pasted onto a planar face. Multi-sketch features such as sweeps and lofts cannot be copied. Likewise, certain applied features such as draft cannot be copied, although fillets and chamfers can.

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.

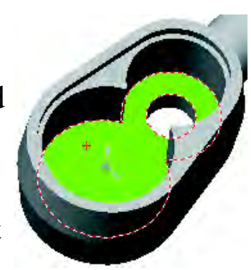
45 Identify the feature to copy.
The feature to be copied must be identified either in the FeatureManager design tree or on the model. For this example, select the feature Wheel Hole by picking it in the FeatureManager design tree. Next, copy it to the clipboard using the **Copy** option on the Standard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



Note

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

46 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.



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

48 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.

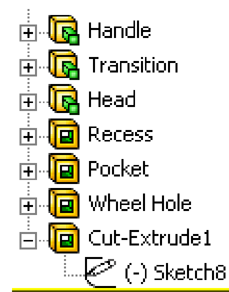
49 Click Delete.

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 8: Editing: Repairs*.

50 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.

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



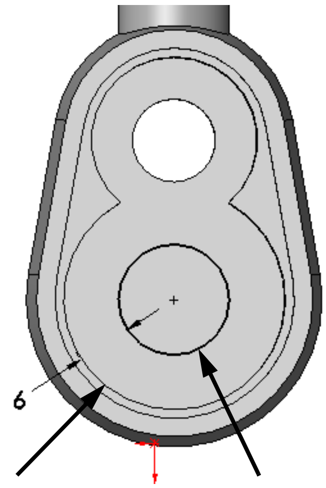
Relate and Change the Sketch

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.

52 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**.

53 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.

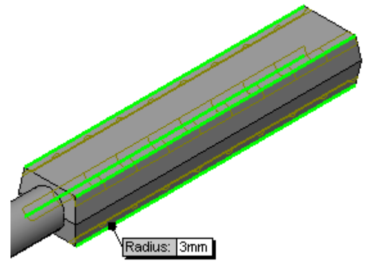
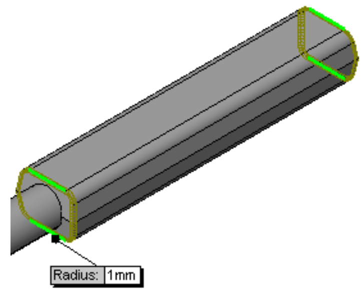
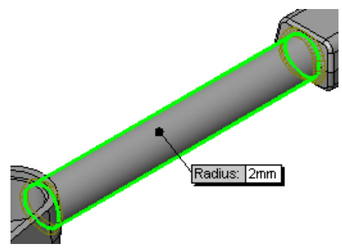
Add a concentric circle dimension by dimensioning the circle and edge. The sketch is now fully defined.

54 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`.

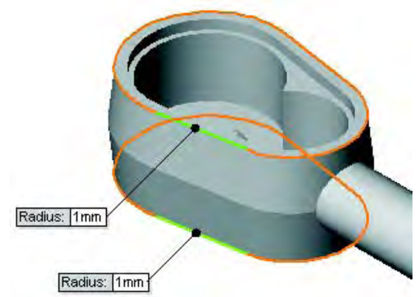
55 Fillets.
Add fillets on edges and faces as shown below.

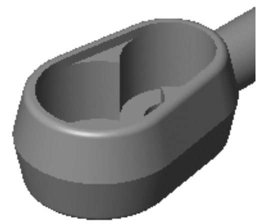
R = **3mm** Name = Handle Fillets	
R = **1mm** Name = H End Fillets	
R = **2mm** Name = T-H Fillets	

Editing the Fillet Feature

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.

56 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.



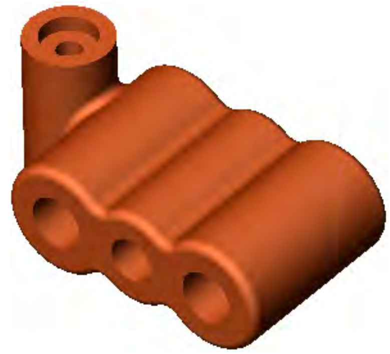


57 Save and close the part.

Exercise 12: Tool Holder

This lab reinforces the following skills:

- *Symmetry in the Sketch* on page 119.
- *Mid Plane Extrusion* on page 121.
- *Introducing: Sketched Circles* on page 124.
- *Trim and Extend* on page 136.
- *Copy and Paste Features* on page 144.



Units: **millimeters**

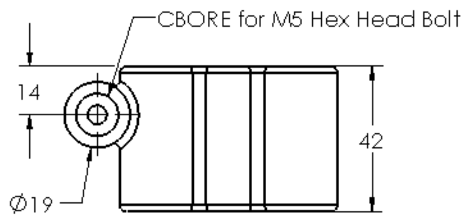
Design Intent

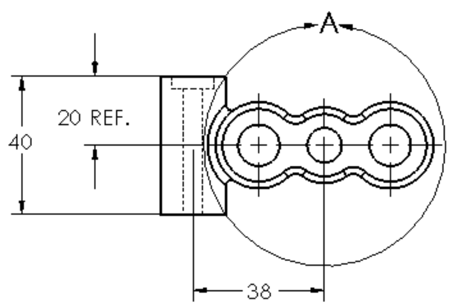
Some aspects of the design intent for this part are:

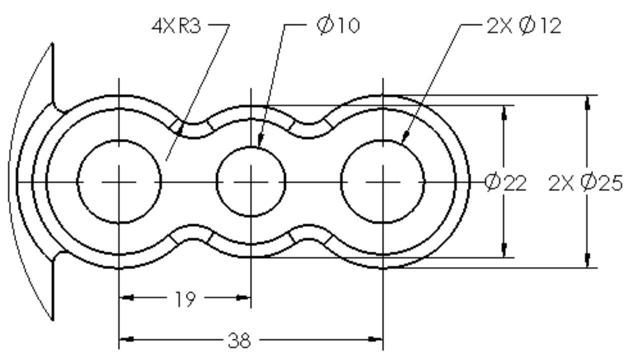
1. All fillets and rounds are **R2mm** unless otherwise noted.
2. Circular edges of equal radii/diameter should remain equal.

Dimensioned Views

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







DETAIL A
SCALE 2 : 1

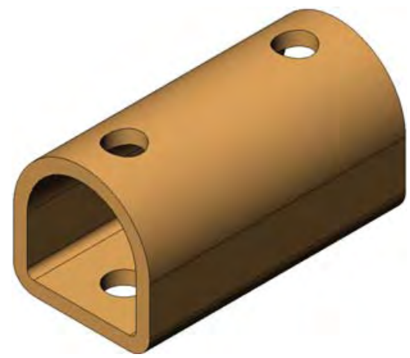
Exercise 13: Symmetry and Offsets 1

Use offsets and symmetry to complete the part.

This lab reinforces the following skills:

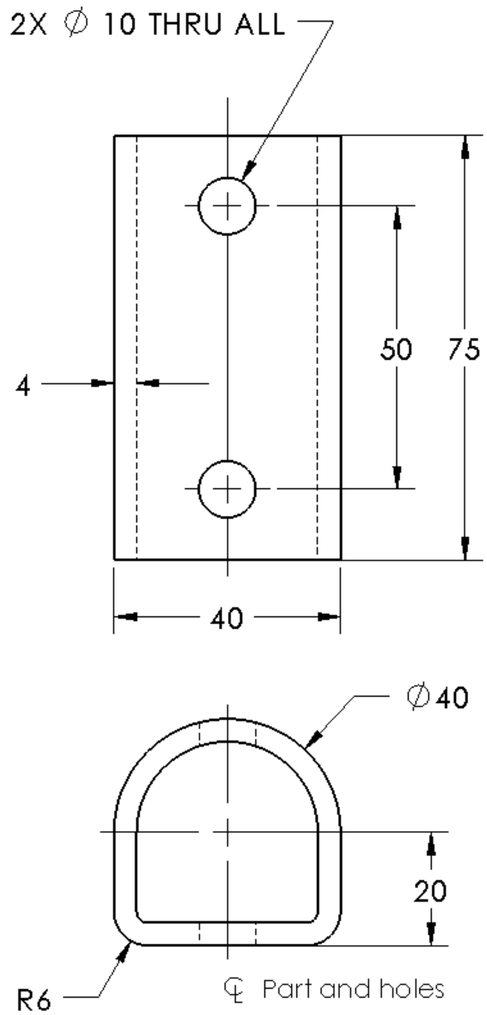
- *Introducing: Insert Centerline* on page 119.
- *Symmetry after Sketching* on page 120.
- *Sketching an Offset* on page 135.

Units: **millimeters**



Dimensioned Views

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

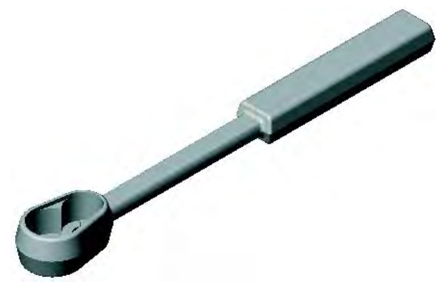


Exercise 14: Ratchet Handle Changes

Make changes to the part created in the previous lesson.

This exercise uses the following skills:

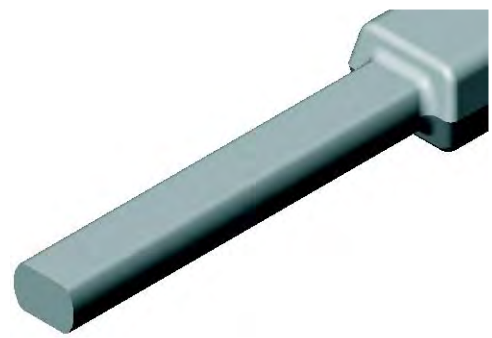
- *Trim and Extend* on page 136.



Design Intent

Some aspects of the design intent for this part are:

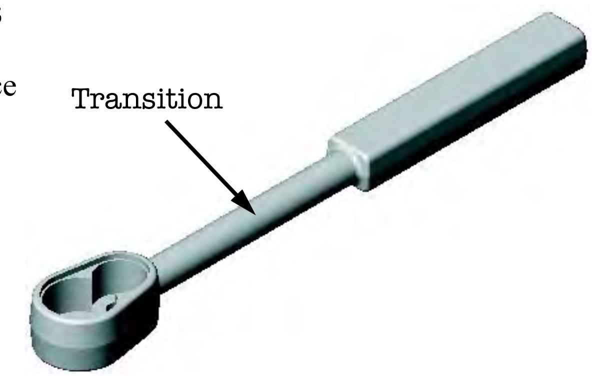
1. The part must remain symmetrical about the Right plane.
2. The Transition requires flats that are driven by the distance between them.



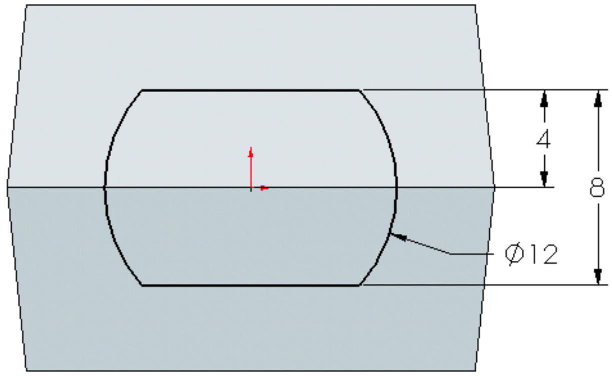
Procedure

Open an existing part.

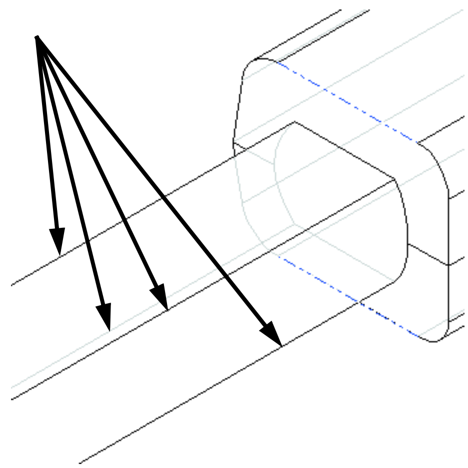
1 Open the part Ratchet Handle Changes.
The change will take place in the shape of the Transition feature.



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.

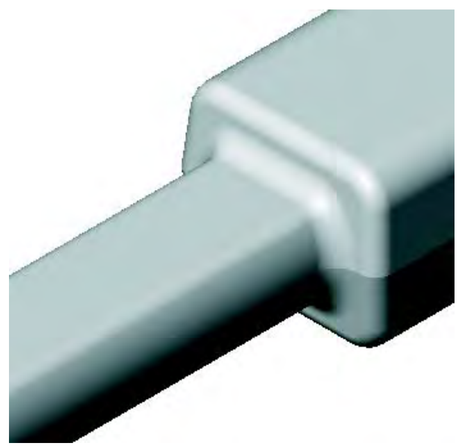


3 Edit Feature.
Edit the `H End Fillets` feature to add more edges. Select the four new edges created by the flats. Click **OK**.



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.

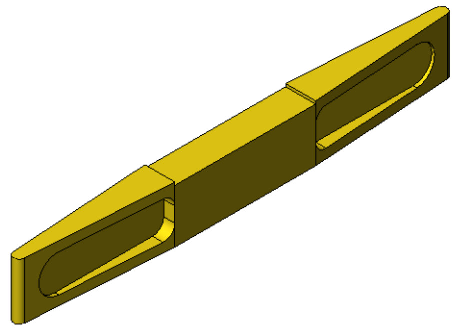


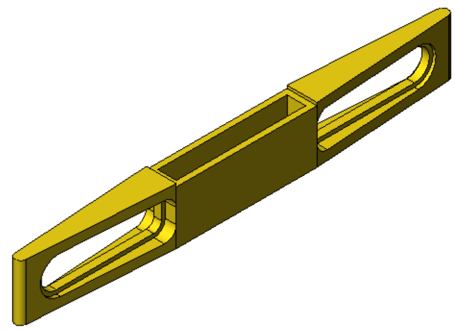
Exercise 15: Symmetry and Offsets 2

Use offsets and symmetry to complete the part.

This lab reinforces the following skills:

- *Introducing: Insert Centerline* on page 119.
- *Symmetry after Sketching* on page 120.
- *Introducing: View Normal To* on page 123.
- *Sketching an Offset* on page 135.
- *Introducing: Offset From Surface* on page 139.
- *Measuring* on page 141.

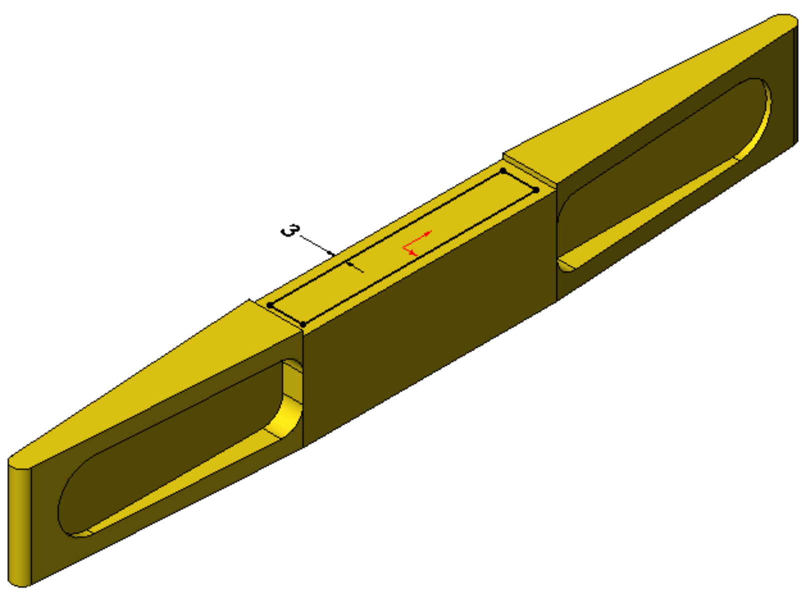




Procedure

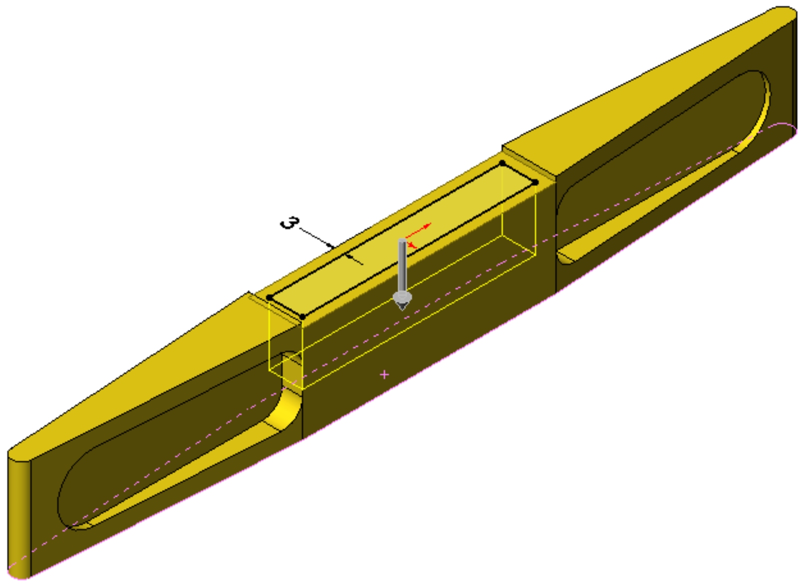
Open the existing part Offset_Entities and create the geometry as shown in the following steps.

1 **Offset entities.**
Use **Offset Entities** to create the sketch geometry shown.

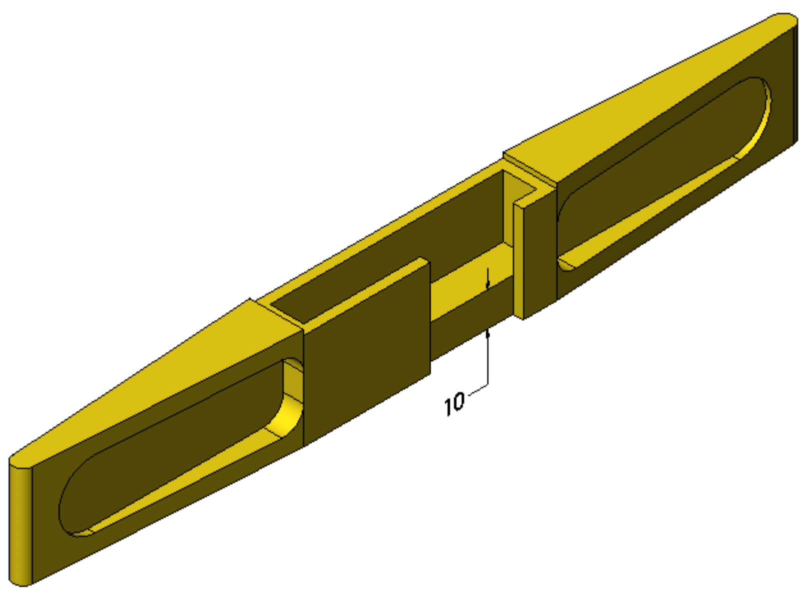


2 Offset from surface.

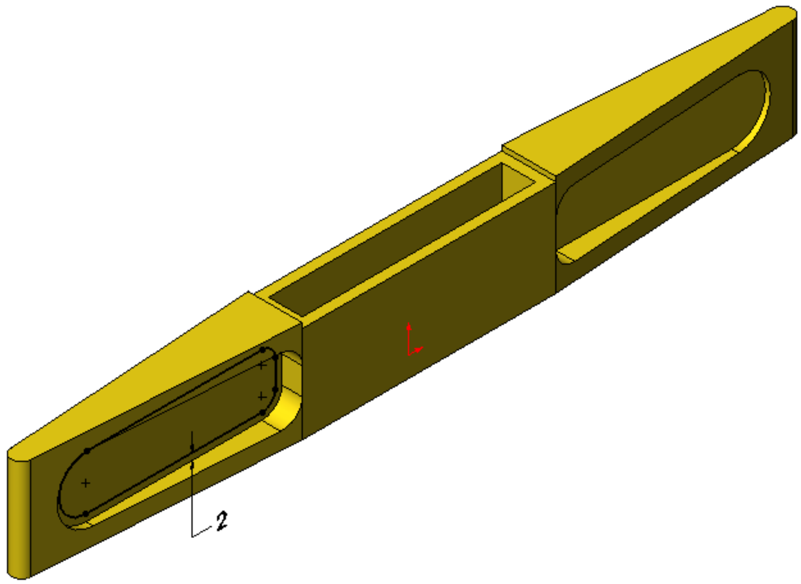
Use a **Cut Extrude** with an **Offset From Surface** of **10mm.** Select the hidden face using **Select Other**.



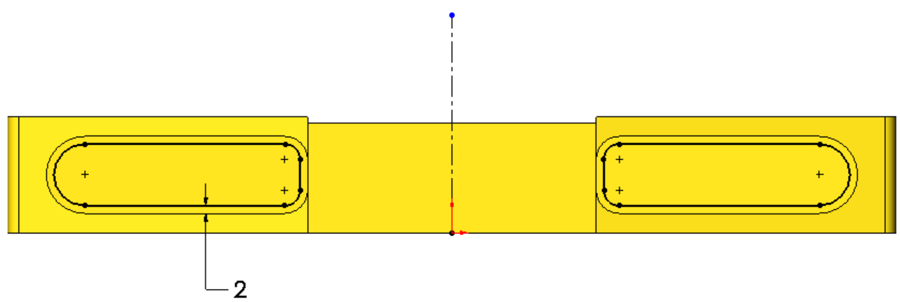
This cutaway shows the results of the offset from surface option. The Measure tool can be used to check this value.



3 Offset.
Create a new sketch and offset the face **2mm**.



4 Offsets with symmetry.
Use **View Normal To** to orient the part. Use **Offset Entities, Center lines** and **Mirror Entities** to create and mirror sketch geometry as shown. Use a **Cut Extrude Through All**.



5 Save and close the part.

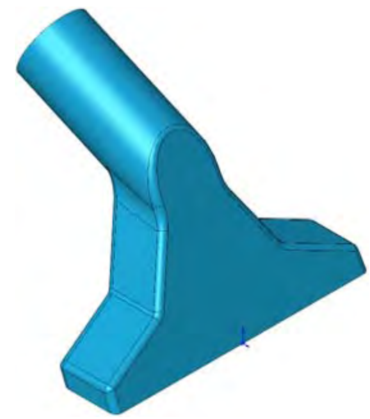
Exercise 16: Up To Surface

Use symmetry and an up to surface end condition to complete the part.

This lab reinforces the following skills:

- *Introducing: Dynamic Mirror* on page 119.
- *Up To Next vs. Up To Surface* on page 126.

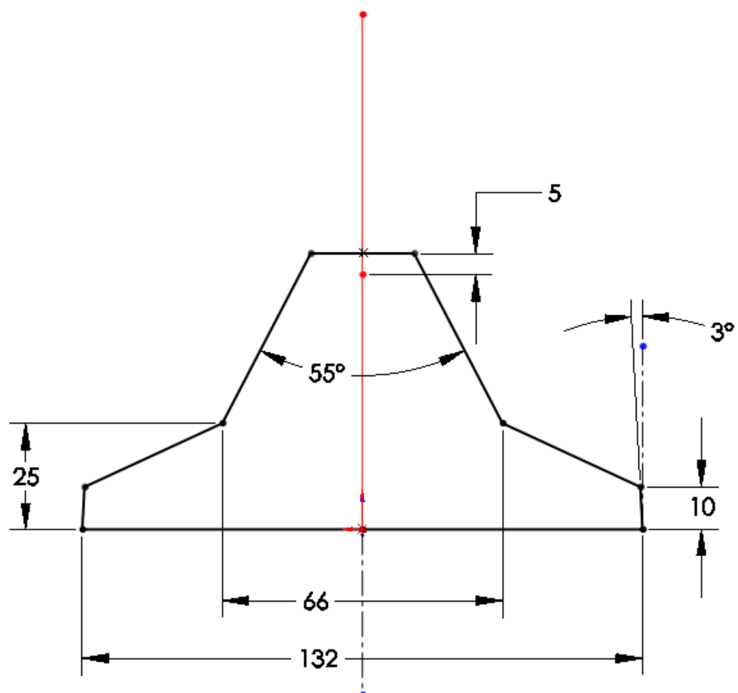
Units: **millimeters**



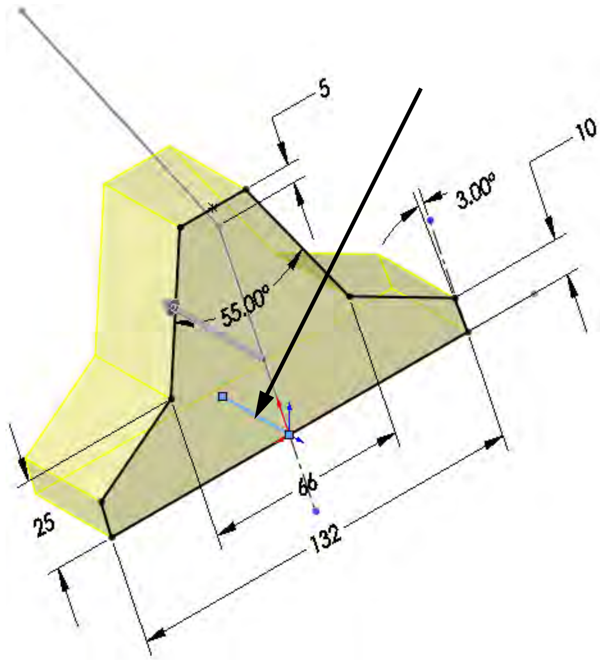
Procedure

Open an existing part named `Up_To_Surface`.

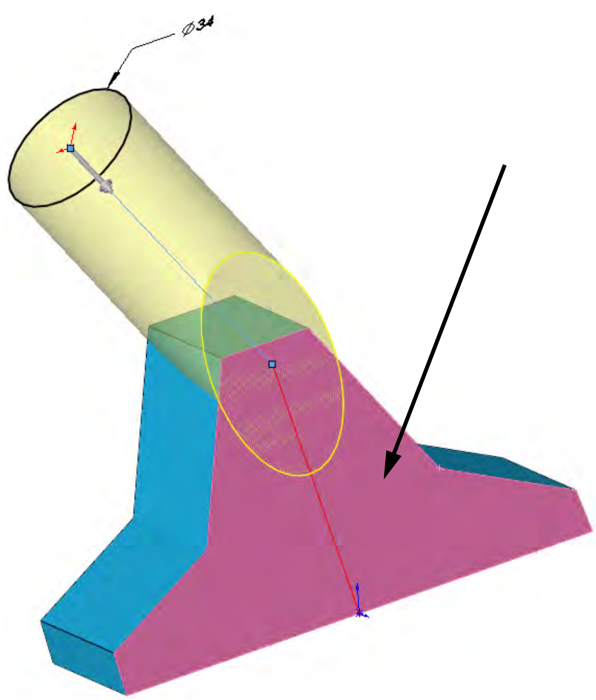
1 Sketch.
Using `Plane2`, create the geometry below using lines and symmetry.



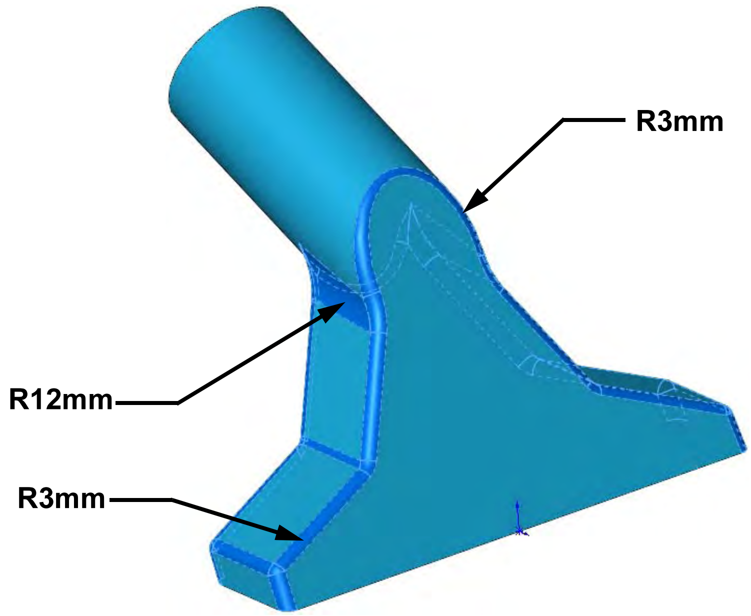
2 Extrude with direction.
Extrude the sketch. Click in the **Direction of Extrusion** field and select the blue sketch line as shown. Set the **Depth** to **28mm**.



3 Up to surface.
Sketch on `Plane1` and create the circle dimensioned as shown. Extrude the sketch **Up To Surface** and select the pink face to terminate the extrusion.



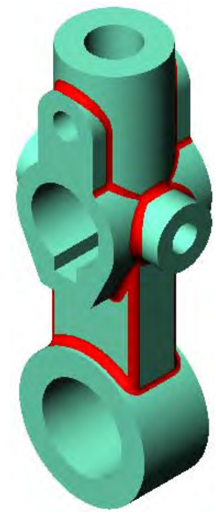
4 Fillets.
Add fillets as shown below.



5 Save and close the part.

Exercise 17: Idler Arm

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



The only reference planes required for this part are `Top`, `Front` and `Right`.

This lab uses the following skills:

- *Symmetry in the Sketch* on page 119.
- *Mid Plane Extrusion* on page 121.
- *Introducing: Sketched Circles* on page 124.
- *Extruding Up To Next* on page 126.
- *Copy and Paste Features* on page 144.

Units: **millimeters**

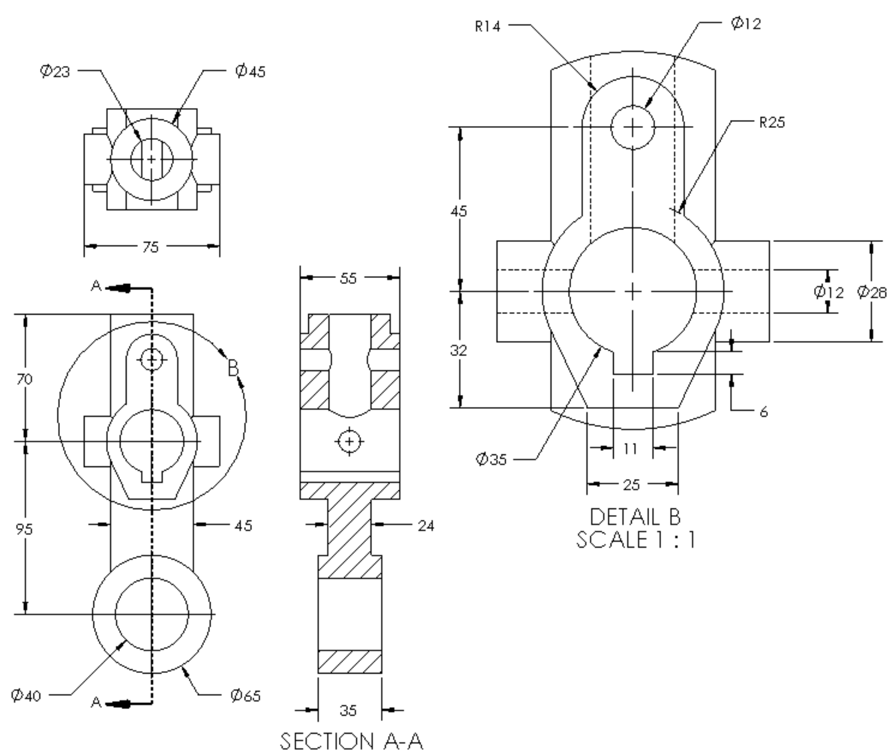
Design Intent

The design intent for this part is as follows:

1. The part is symmetrical.
2. Front holes are on the centerline.
3. All fillets and rounds (highlighted red above) are **R3mm** unless noted.
4. Center holes in `Front` and `Right` share a common centerpoint.

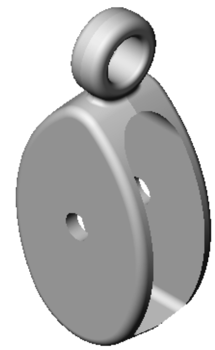
Dimensioned Views

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



Exercise 18: Pulley

Complete this part using the dimensions provided. Use relations where applicable to maintain the design intent.

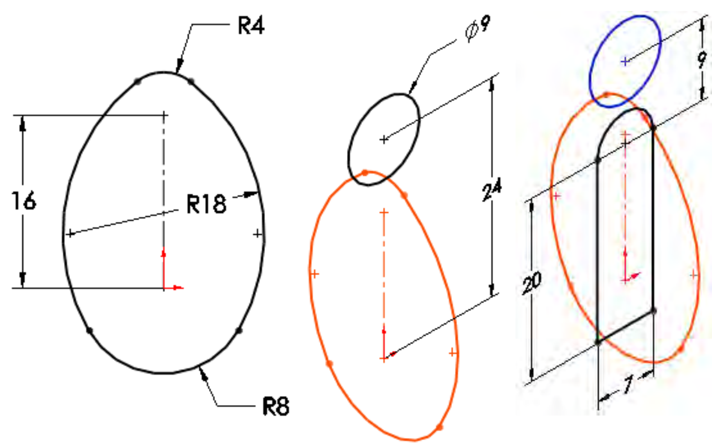


This lab uses the following skills:

- *Symmetry in the Sketch* on page 119.
- *Mid Plane Extrusion* on page 121.
- *Draft Toggle* on page 121.
- *Introducing: Sketched Circles* on page 124.

Optional Sketching

If you would like to use the existing geometry, skip to **Procedure**. If you would like to create the sketches, open a new **mm** part and use the dimensions below. There are three sketches required, the first created on the Front Plane, the remaining two on the Right Plane.

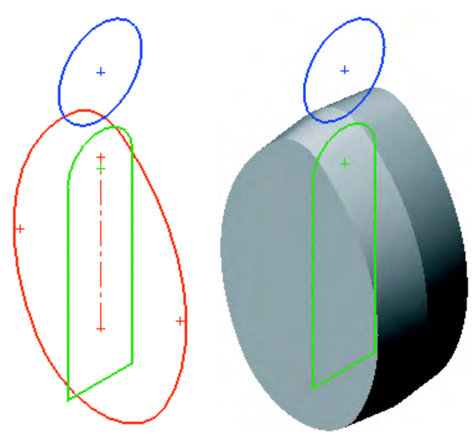


Front Plane Right Plane

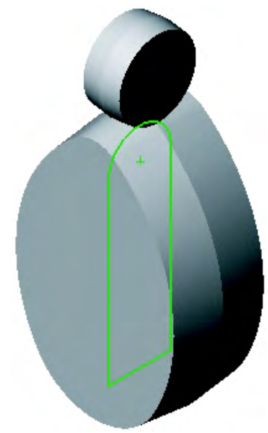
Procedure

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.

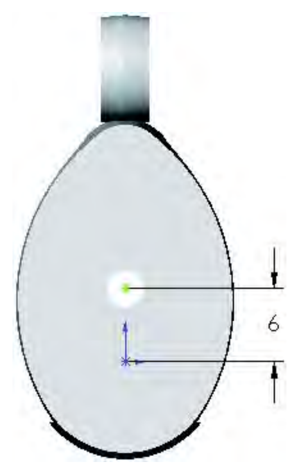


3 Cut and hole.
Create a cut using the `Center Cut` sketch (green). The cut is **Through All** in both directions.

Add a **5mm** diameter hole.

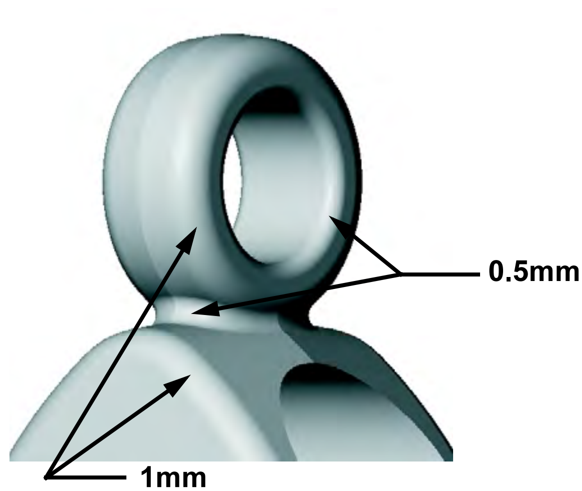
Add the fillet (**1mm**) to the bottom edges after the cut.

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



4 Fillets.
Add fillets of **1mm** and **0.5mm** 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

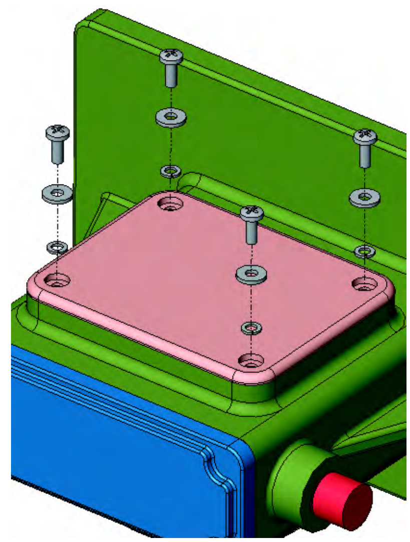
Upon successful completion of this lesson, you will be able to:

- Create a linear pattern.
- Add a circular pattern.
- Use geometry patterns properly.
- Create a mirror pattern.
- Use the pattern seed only option with a linear pattern.
- Add a sketch driven pattern.
- Automate the process of fully defining a sketch.

Why Use Patterns?

Patterns are the best method for 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 of patterns is to support the use of Smart Fasteners. Smart Fasteners are used 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 the following table is intended to highlight the typical uses for each type.

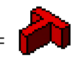
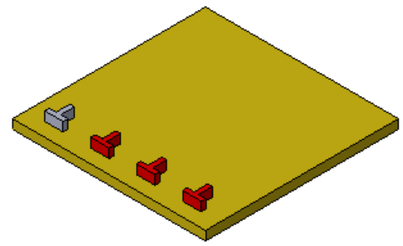
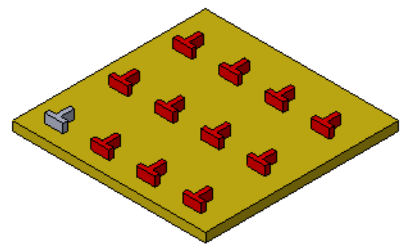
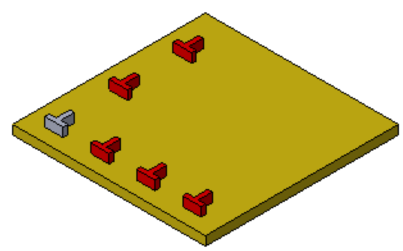
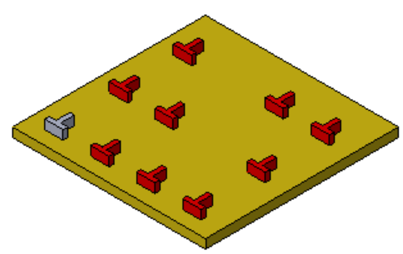
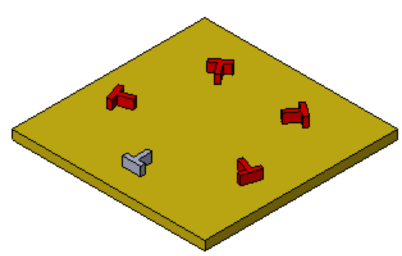
Note

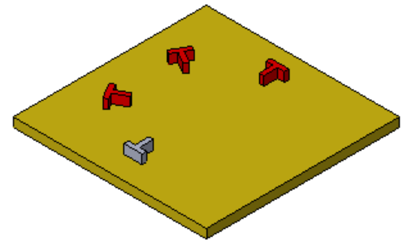
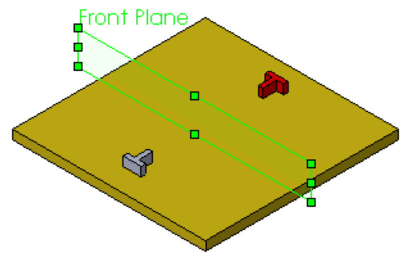
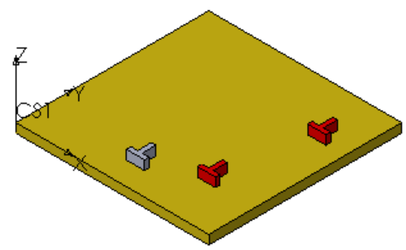
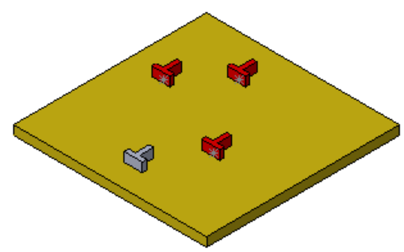
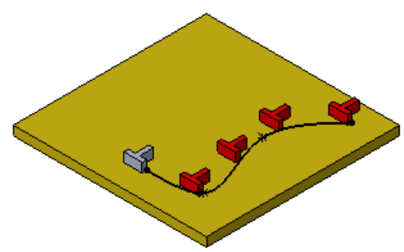
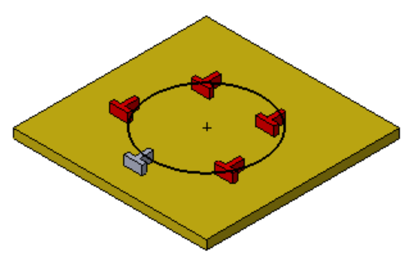
This table lists all types of patterns; not all of them are shown as case studies.

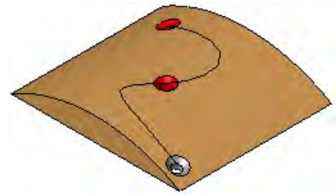
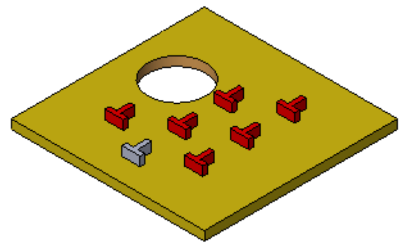
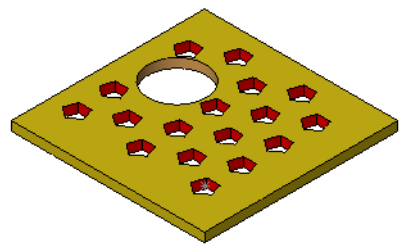
- **Seed**
  The seed is the geometry to be patterned. It can be one or more features, bodies or faces.
- **Pattern Instance**
  The **Pattern Instance** (or just **Instance)** is the “copy” of the seed created by the pattern. It is in fact much more than a copy because it is derived from the seed and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

Types of Patterns

The pattern types and how they are typically used are listed in the table below.

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	✓	✓			✓		

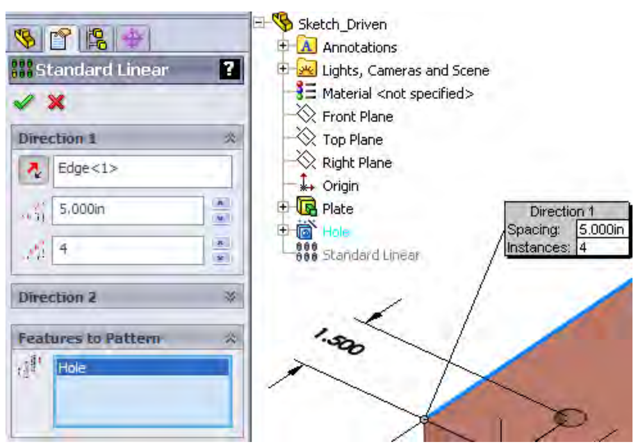
Pattern Feature	Select Feature, Bodies or Faces	Propagate Visual Properties	Pattern Seed Only	Skip Instances	Geometry Pattern	Vary Sketch	Use Shapes
Sketch Driven	✓	✓			✓		
Curve Driven	✓	✓	✓	✓	✓	✓	
Fill	Features and Faces only	✓		✓	✓	✓	✓

Note

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

Flyout FeatureManager Design Tree

The flyout FeatureManager design tree enables you to view both the FeatureManager design tree and the PropertyManager at the same time. This enables you to select features from the FeatureManager design tree 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.

Reference Geometry

There are two types of **Reference Geometry** that are useful in creating patterns: **Temporary Axes** and **Axes**.

Temporary Axes

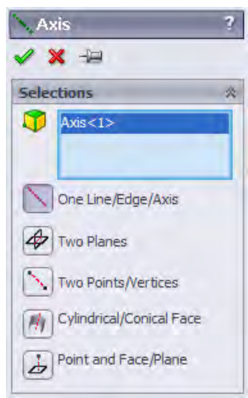
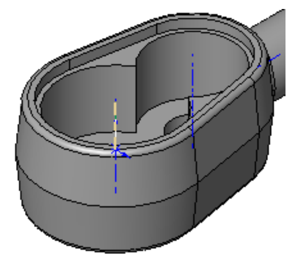
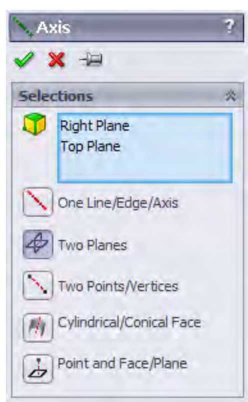
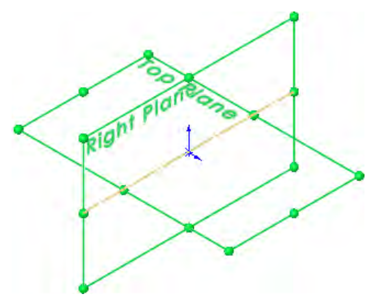
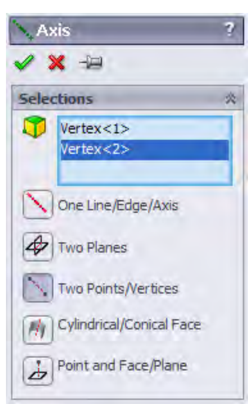
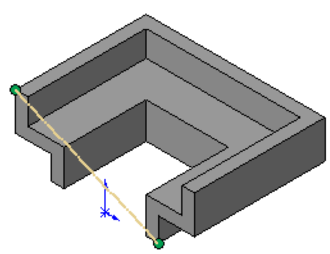
Every cylindrical and conical feature has an axis associated with it. View the temporary axes of the part using **View, Temporary Axes**. One axis is displayed through each circular face in the model.

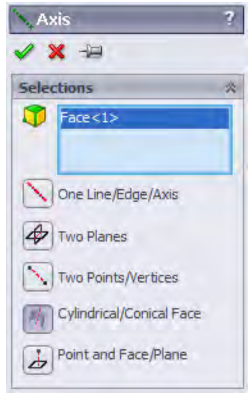
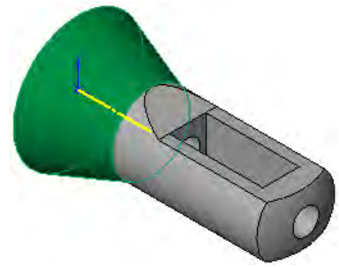
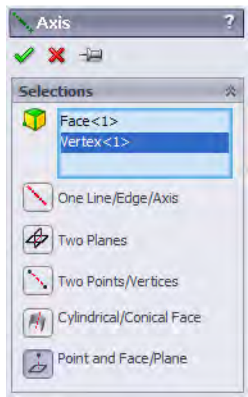
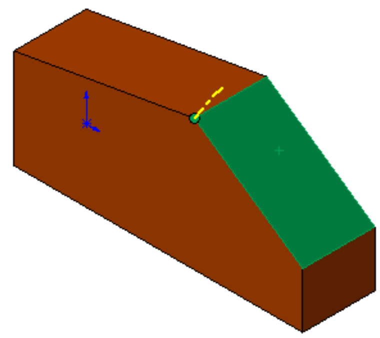
Axes

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 design tree, and resized.

Where to Find It

- Click **Axis** on the Reference Geometry toolbar.
- Or, click **Insert, Reference Geometry, Axis...**.

Temporary Axes can be made permanent and given unique names using the **One Line/Edge/Axis** option.		
Select two planes or planar faces and the option **Two Planes**.		
Select **Two Points/Vertices** to define an axis through them.		

<table>
<tr>
<td>Select a Cylindrical/ Conical Face to define an axis through the rotational center.</td>
<td>
</td>
<td></td>
</tr>
<tr>
<td>Select a plane or planar face and a point or vertex to define an axis normal to the plane through the point.</td>
<td>
</td>
<td></td>
</tr>
</table>

Linear Pattern

The **Linear Pattern** tool 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.

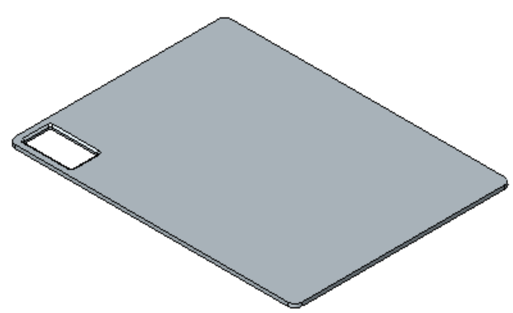
Introducing: Linear Pattern

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

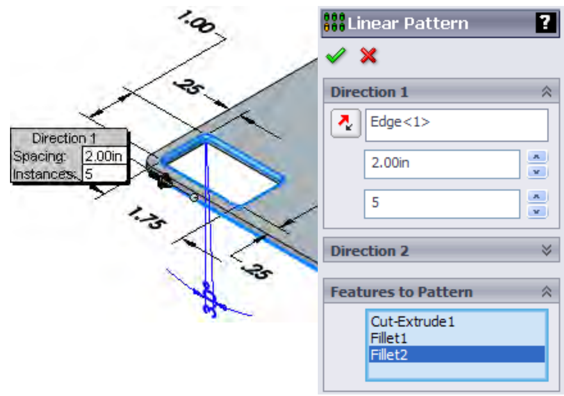
Where to Find It

- On the Features toolbar, click the **Linear Pattern** tool from the **Pattern** flyout 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**.

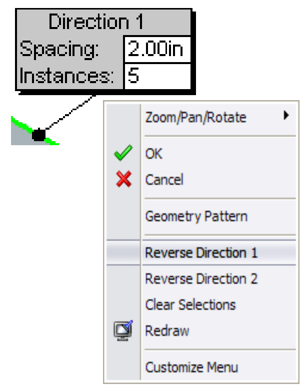


Select the linear edge of the part and click the **Reverse Direction**, if necessary, to set the direction shown. Set the **Spacing** to **2”** and **Number of Instances** to **5**.

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

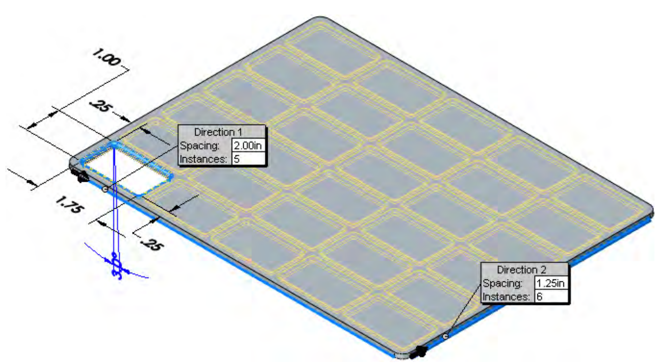
Note

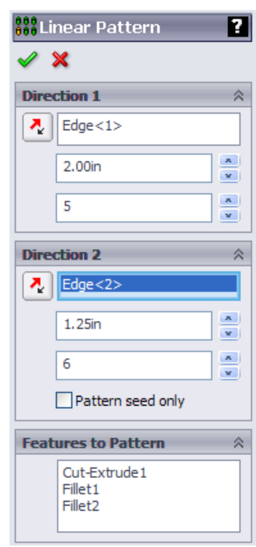
The callout is attached to the geometry used to define the pattern direction or axis. It contains the key settings for **Spacing** and **Instances** and is editable. Double-click the setting to change and retype the value.



Right-click the callout 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. Set the **Spacing** and **Instances** as shown.



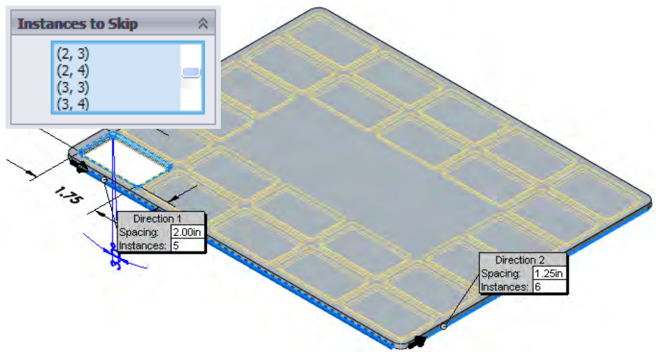


Deleting Instances

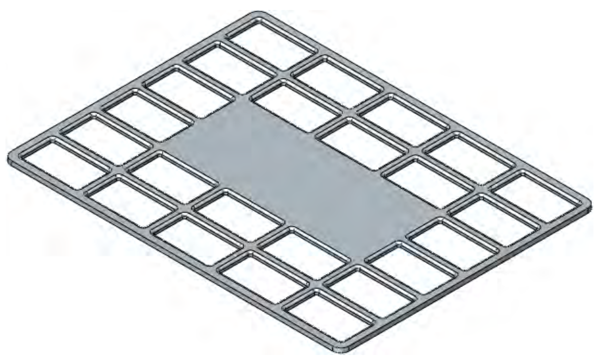
Instances that are generated by the pattern can be deleted by selecting a marker at the centroid of the instance shown in the pattern preview. Each instance is listed in array format **(2,3)** for identification.

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.



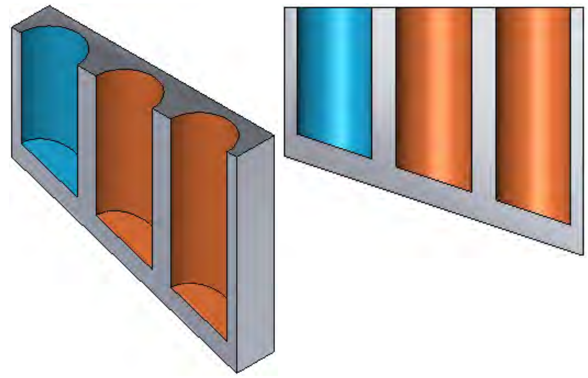
5 **Completed pattern.**
Click **OK** to add the pattern feature `LPattern1`.



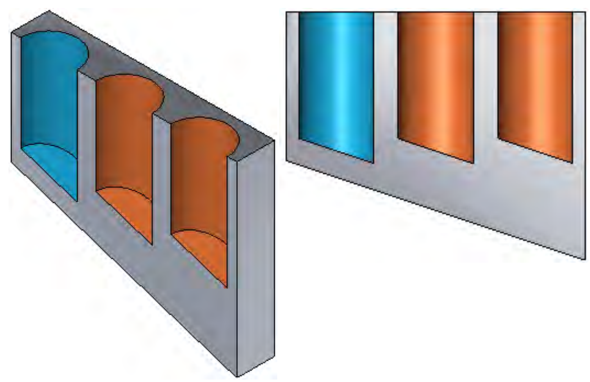
Geometry Patterns

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.

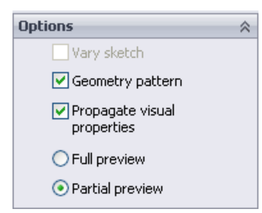
- **Without Geometry Pattern**
  If the **Geometry Pattern** option is *cleared*, the end condition of the seed is used in the instances. In this example, the **Offset From Surface** end condition of the blue seed feature is applied in the orange instances, forcing them to use the same end condition.



- **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 and click **OK**. Because the plate is constant thickness, the resulting geometry will look the same.



7 **Save and close the part.**

Circular Patterns

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

Introducing: Circular Pattern

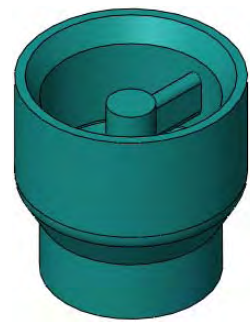
Circular Pattern creates multiple instances of one or more features spaced around an axis. The axis can be an circular face, edge, axis, temporary axis or angular dimension.

Where to Find It

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

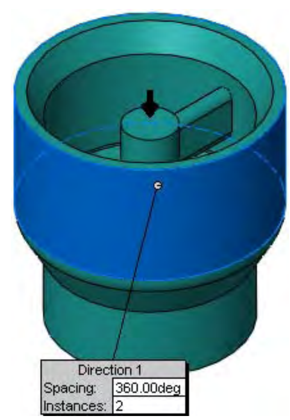
---

1 Open the part named Circular_Pattern.



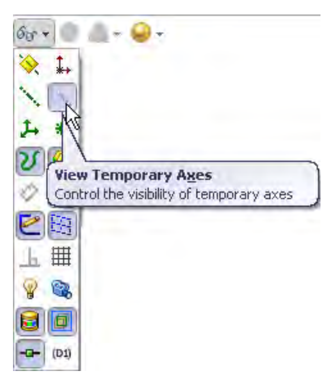
2 Pattern Axis.
Click **Insert, Pattern/Mirror, Circular Pattern...**.

Click in **Pattern Axis** and select the cylindrical face of the model as shown.



Tip

To use a temporary axis as the pattern axis, click **View, Temporary Axes** or click the on-screen menu option.

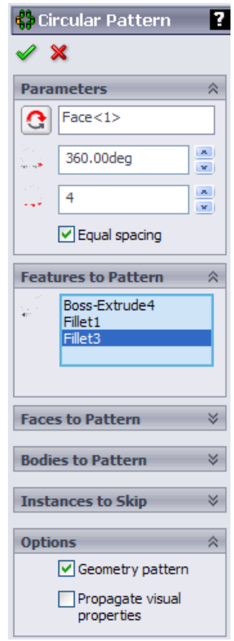


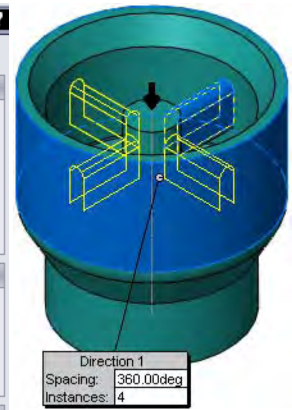
3 Settings.

Click in **Features to Pattern** and select the three features shown for **Features to Pattern**.

Click **Equal Spacing**, **4** instances and click **Geometry pattern**.

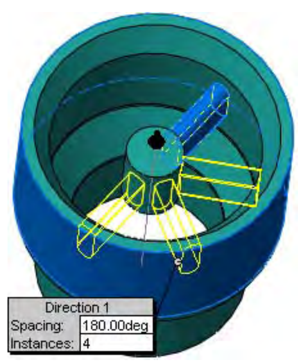
Check that the **Angle** is set to **360°** and click **OK**.

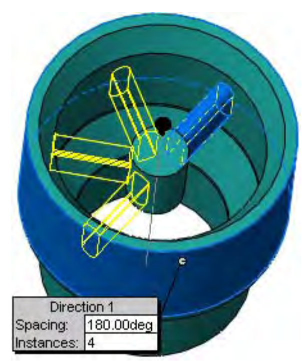




Note

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





4 Save and close the part.

Mirror Patterns

The **Mirror Pattern** tool creates a copy, or instance, across a plane or planar face. The instance is dependent on the original. Changes to the original propagate down to the mirrored instance(s).

Introducing: Mirror Pattern

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

Where to Find It

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

Patterning a Solid Body

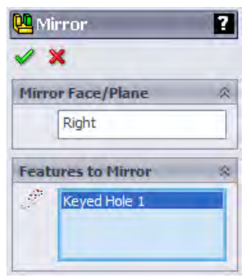
To mirror all the geometry of a part about a common face, select the common face as **Mirror Face/Plane** and the solid body as **Bodies to Mirror**. The common face must be planar.

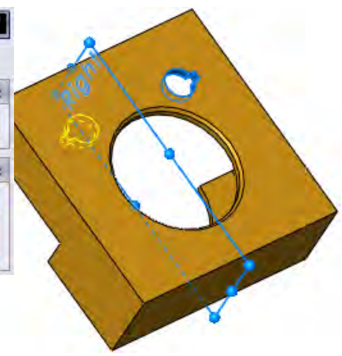
---

1 **Open the part named Mirror_Pattern.**

2 **Mirror.**
Click **Insert**, **Pattern/ Mirror, Mirror** and the Right plane. Select the library feature Keyed Hole 1 as the **Features to Mirror**.

Click **OK**.

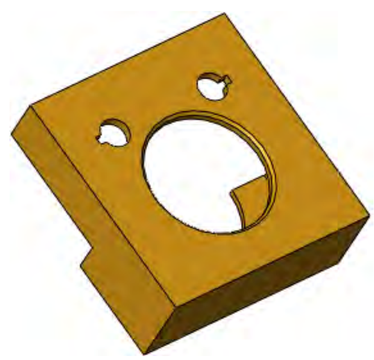




Note

Geometry Pattern can also be used with this feature.

3 **Save and close the part.**



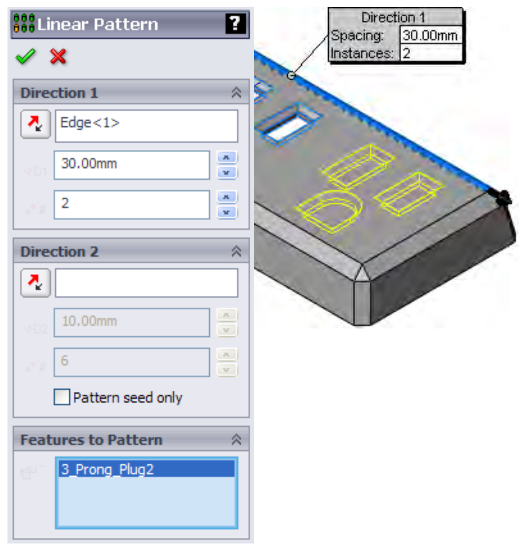
Using Pattern Seed Only

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** and **30mm** as the **Spacing**, **2** as the **Number of Instances**.

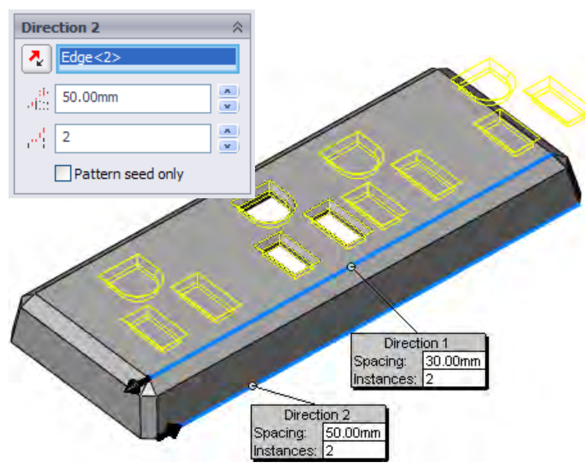
For **Features to Pattern**, select the library feature 3_Prong_Plug2.



Note

An existing pattern feature can be used as the **Features to Pattern**. This enables you to pattern the pattern.

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**.

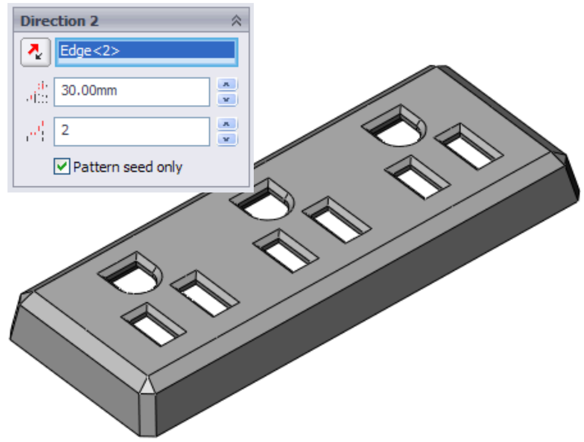


Note

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**.

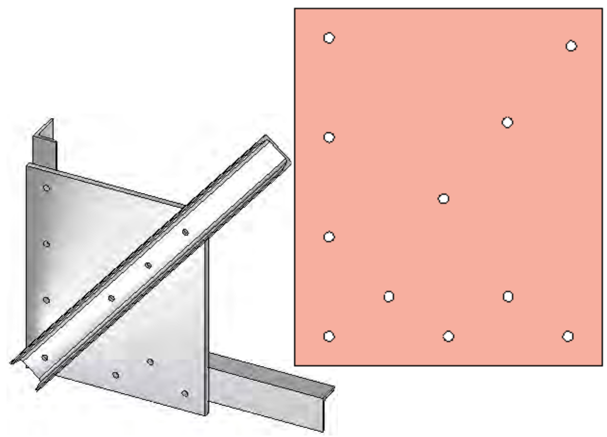


5 **Save and close the part.**

Sketch Driven Patterns

The **Sketch Driven Patterns** tool creates copies, or instances, in a linear arrangement controlled by sketch points. The pattern can be based on the centroid of the seed or a selected point off the centroid.

This example represents the holes in a structural steel plate.



Note

This pattern is intended for use where a linear type of pattern is required but the standard **Linear Pattern** feature can not easily be utilized.

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.

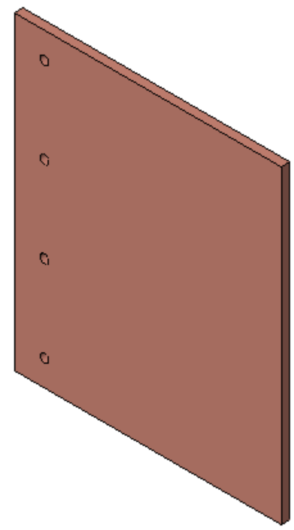
Where to Find It

- 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 to position points but will be ignored by the pattern.

1 Open Sketch_Driven.
The part contains a seed feature (Hole) and an existing linear pattern feature (Standard Linear).



Introducing: Point

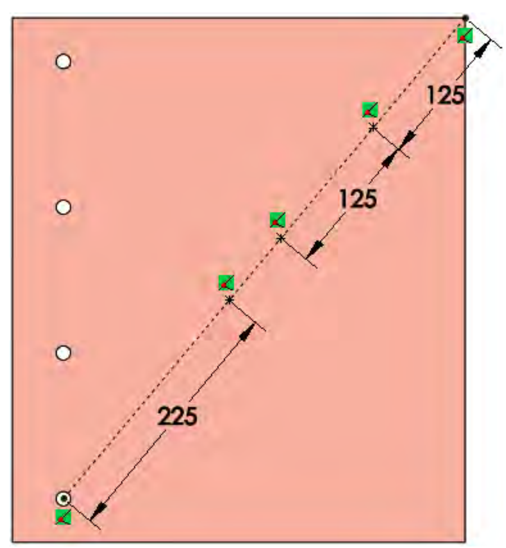
The **Point** tool creates point entities in the active sketch. The sketch entity **Point** can be used to locate a position in a sketch that other geometry (endpoints for example) cannot.

Where to Find It

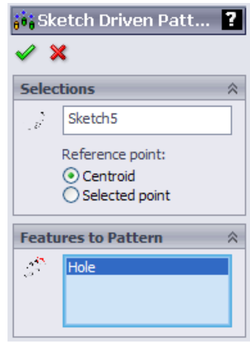
- On the Sketch toolbar click the **Point** tool.
- From the **Tools** menu choose: **Sketch Entities, Point**.

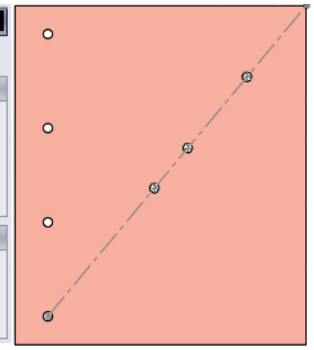
2 Sketch with points.
Open a new sketch on the top face of the Plate feature. Create the centerline and add the points and dimensions as shown.

Close the sketch.

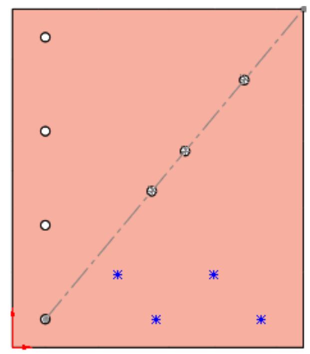


3 Sketch driven pattern.
Click **Sketch Driven Pattern** and select the new sketch and the **Centroid** option. Under **Features to Pattern**, select the Hole feature and click **OK**.





4 Add points.
Create another sketch and add points in the pattern shown, using inferencing to line up the rows horizontally as shown.



Note

Points cannot be added directly to existing sketch endpoints. The message

`Sketch points cannot be added at the same location as an existing point.`

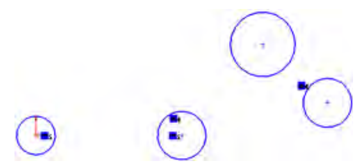
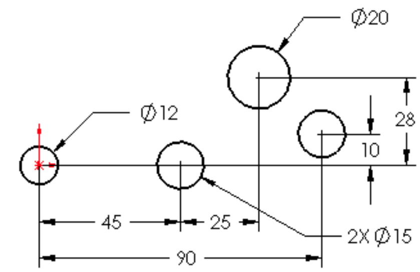
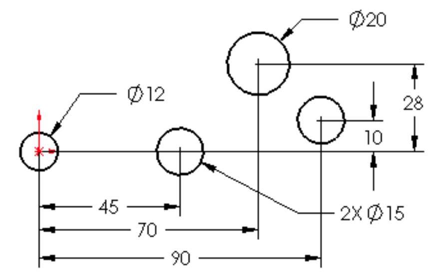
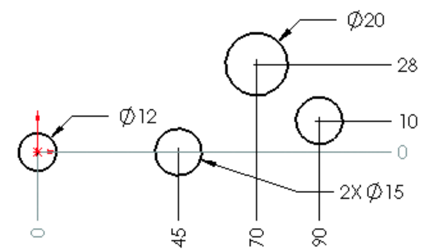
will appear if you try. Instead, place the points off the geometry and drag and drop them onto the endpoints later.

Automatic Dimensioning of Sketches

Fully Define Sketch creates relations and 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.

Introducing: Fully Define Sketch

Fully Define Sketch has options for dimension type, entities to be dimensioned and starting points.

Under defined sketch with geometric relations.	
Chain option selected with start point at origin. **Note:** Some dimensions have been moved for clarity.	
Baseline option selected with start points at origin.	
Ordinate option selected with start points at origin.	

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, Fully Define Sketch...**.
- Or, on the Dimensions/Relations toolbar, click **Fully Define Sketch** .
- Or, right-click in the sketch, and choose **Fully Define Sketch**.

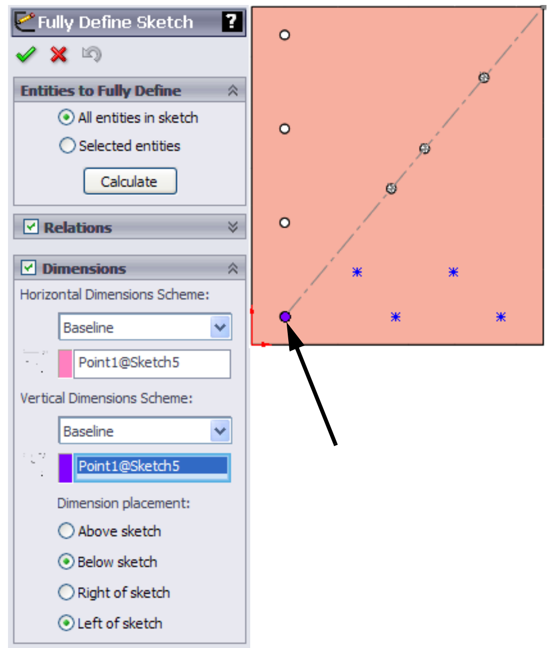
5 Relation and dimension setup.
Click the **Fully Define Sketch** tool.

Leave the **Relations** as default, **Select all**.

In **Dimensions,** select the endpoint of the sketch centerline as the datum for dimensions in both directions.

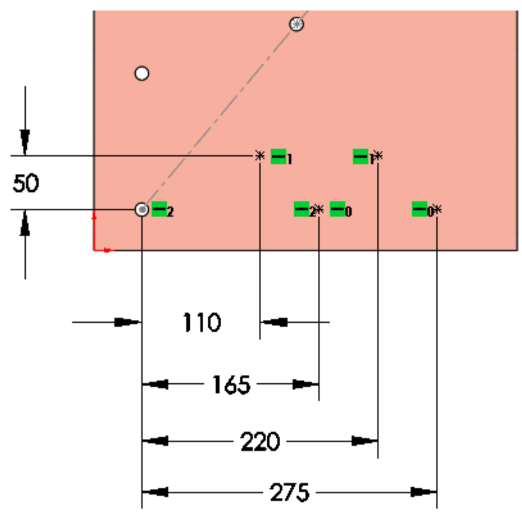
Set both **Schemes** to **Baseline**.

Click **Calculate** and **OK**.



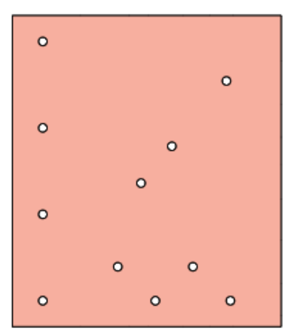
6 Relations and dimensions.
Horizontal relations and dimensions are added to fully define the sketch.

Set the values as shown and close the sketch.



Note Sketches dimensioned this way are fully defined but can be edited. You can delete and replace dimensions if required.

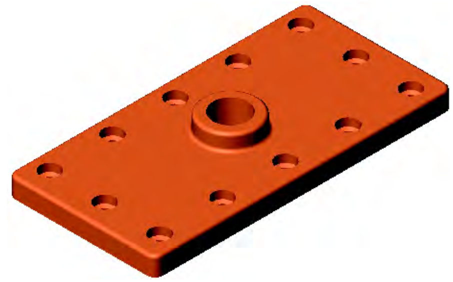
7 Pattern.
Add another sketch driven pattern using the new sketch and the same seed feature, Hole.



8 Save and close the part.

Exercise 19: Linear Patterns

Create feature patterns in this part using a Linear Pattern.



This lab uses the following skills:

- *Linear Pattern* on page 170.
- *Deleting Instances* on page 172.

Procedure

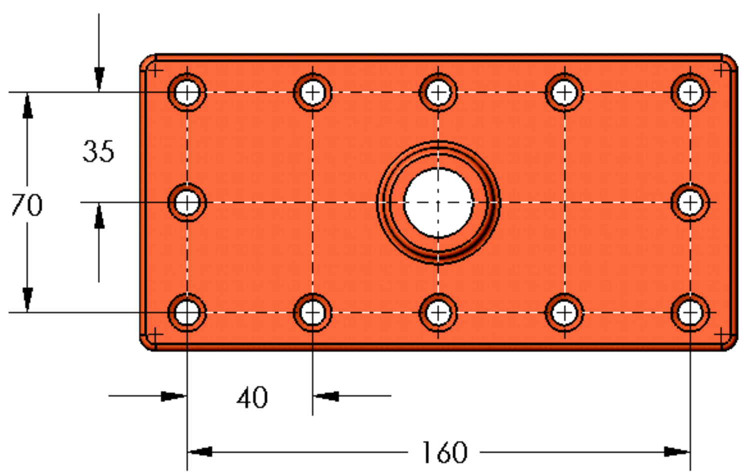
Open an existing part.

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.

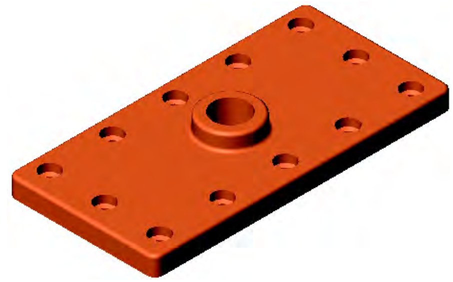
2 Linear pattern.
Create a pattern using the seed. Use the dimensions below.



3 Save and close the part.

Exercise 20: Sketch Driven Patterns

Create feature patterns in this part using a Sketch Driven Pattern.



This lab uses the following skills:

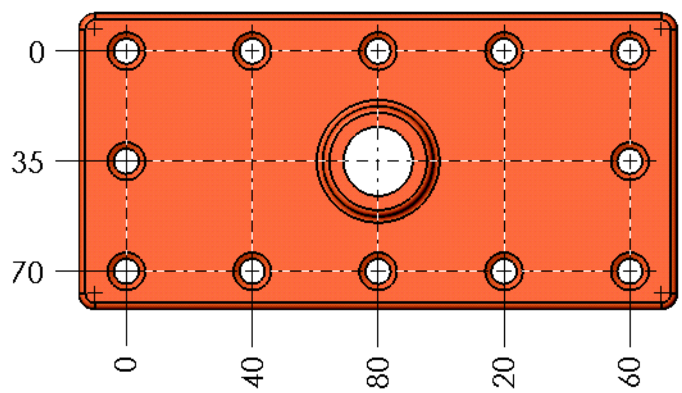
- *Sketch Driven Patterns* on page 178.

Procedure

Open an existing part.

1 Open the part Sketch Driven Pattern.
The part includes the "seed" feature used in the patterns.

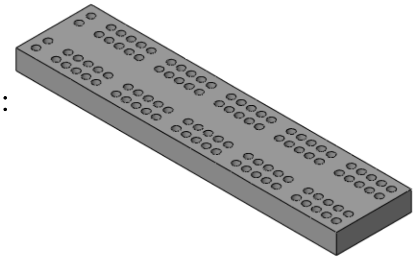
2 Sketch driven pattern.
Use the dimensions below to define the sketch used with the Sketch Driven Pattern.



3 Save and close the part.

Exercise 21: Skipping Instances

Complete this part using the information and dimensions provided.



This lab reinforces the following skills:

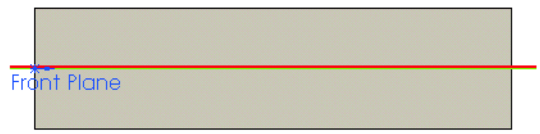
- *Linear Pattern* on page 170.
- *Deleting Instances* on page 172.

Units: **millimeters**

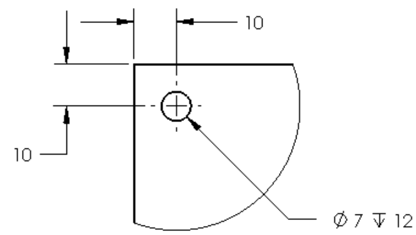
Procedure

Create a new part.

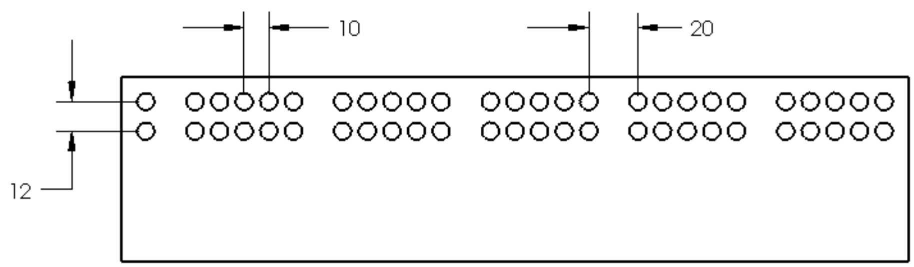
1 **Base feature.**
Create a block **75mmx320mmx20mm**. It will be useful to have a plane centered along the long direction.



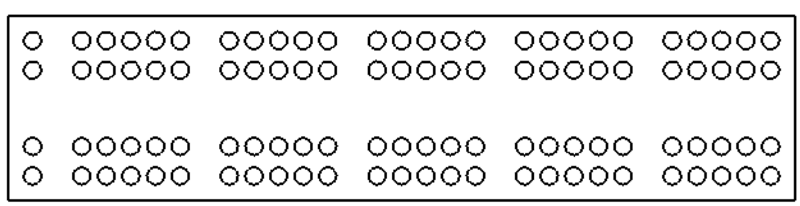
2 **Seed.**
Create the seed feature using the Hole Wizard and an ANSI MM drill.



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 **8mm** diameter and rebuild.

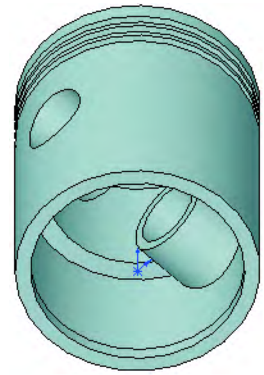
6 **Save and close the part.**

Exercise 22: Linear and Mirror Patterns

Complete this part using the information and dimensions provided.

This lab reinforces the following skills:

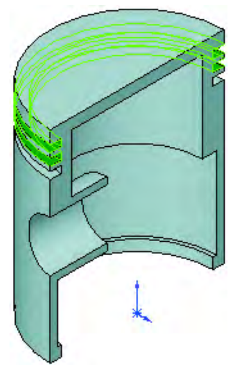
- *Linear Pattern* on page 170.
- *Mirror Patterns* on page 176.
- *Patterning a Solid Body* on page 176.



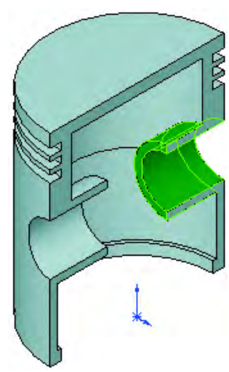
Procedure

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”**.

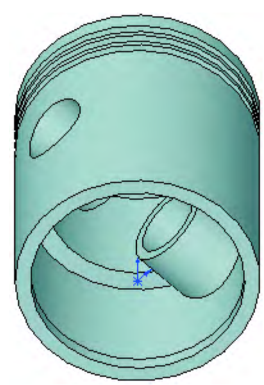


2 **Mirror features.**
Using a single pattern feature, create the duplicate boss and cut as shown.



3 **Symmetry.**
Use a third pattern feature to create the full model from the half model using **Bodies to Mirror**.

4 **Save and close the part.**

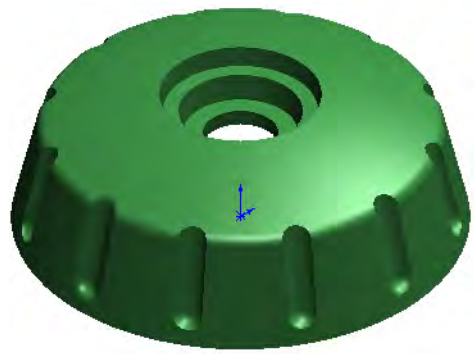


Exercise 23: Circular Patterns

Complete this part using the information and dimensions provided.

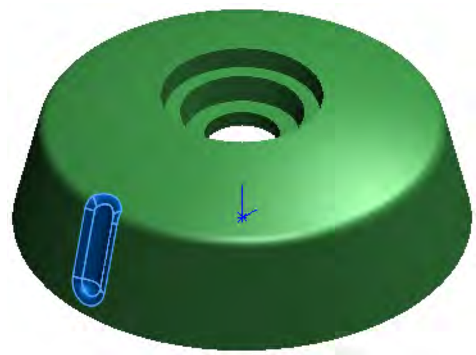
This lab reinforces the following skills:

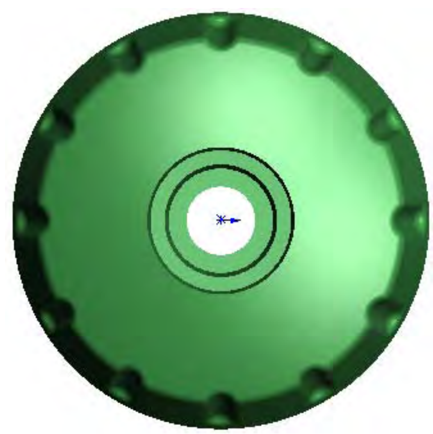
- *Circular Patterns* on page 174.



Procedure

Open the existing part Circular. Use an equally spaced circular pattern to pattern the cut and fillet for 12 total instances.





Lesson 6
Revolved Features

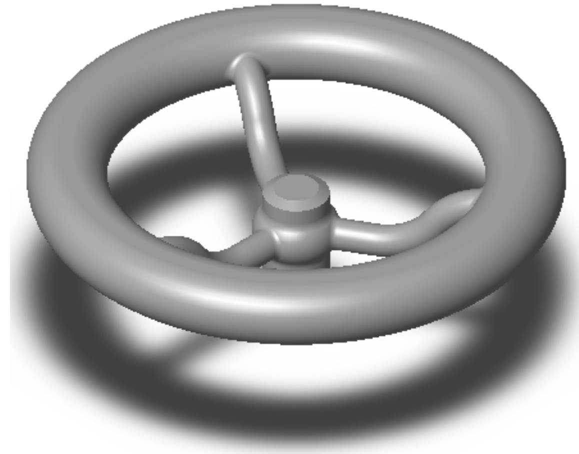
Upon successful completion of this lesson, you will be able to:

- Create revolved features.
- Apply special dimensioning techniques to sketches for revolved features.
- Use the multibody solid technique.
- Create a sweep feature.
- Calculate the physical properties of a part.
- Perform rudimentary, first pass stress analysis.

Case Study: Handwheel

The handwheel requires the creation of revolved features, circular patterns and sweep features.

Also included in this lesson are some basic analysis tools.



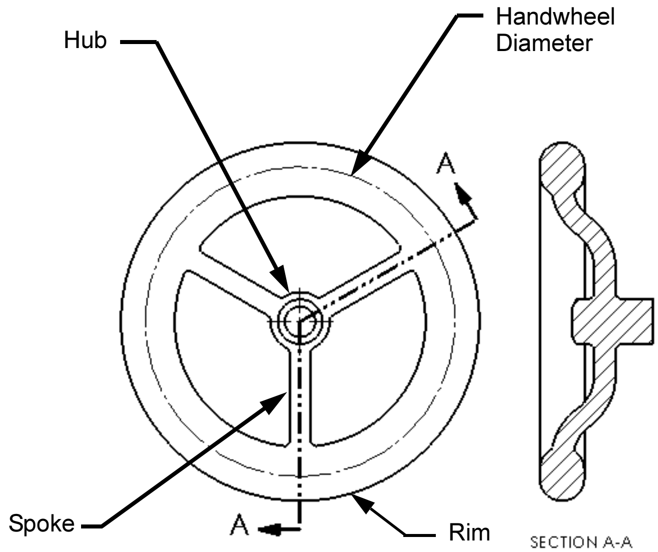
Stages in the Process

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.
- **Analysis**
  Using analysis tools, 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 line (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.

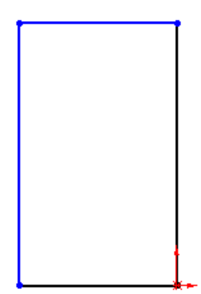
Sketch Geometry of the Revolved Feature

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.

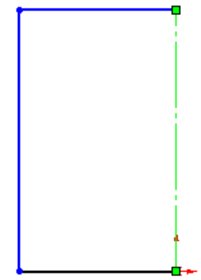
2 Rectangle.

Right-click the `Right Plane` and select **Sketch**.

Create a rectangle from the Origin approximately **50mm** high by **30mm** wide.



3 Convert to construction.
Select the vertical line shown and click **For Construction**. The line is converted into a construction line.



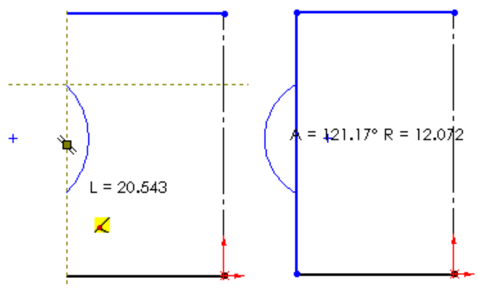
Introducing: 3 Point Arc

The **3 Point Arc** option enables you to create an arc based on three points, the two endpoints followed by a point on the curve.

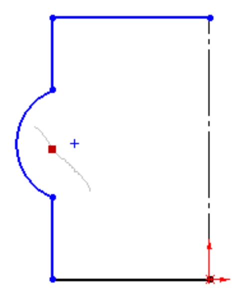
Where to Find It

- 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.



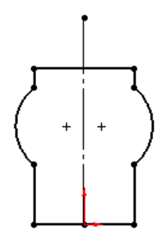
5 Trimming.
Use the **Trim** tool with the **Power Trim** option and trim away the portion of the line inside the arc.



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, axis or sketch line must be specified as the axis of revolution.
- The sketch must not cross the axis.

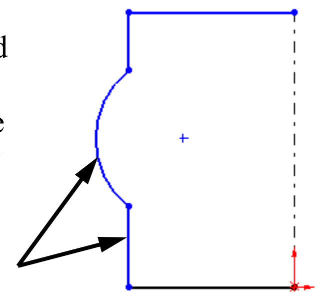

Not Valid

Note that in this example, the right vertical sketch line could be used as the axis of revolution.

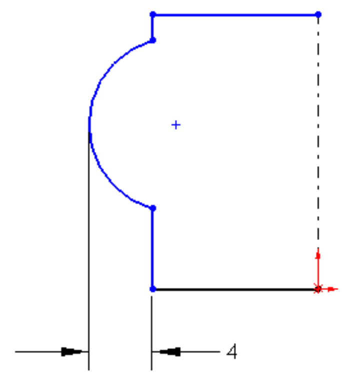
Dimensioning the Sketch

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 the vertical and then **Shift**-selecting the circumference of the arc. The result is a dimension between the line and the tangent of the arc. Using the **Shift** key selects the edge rather than the center of the circle or arc.

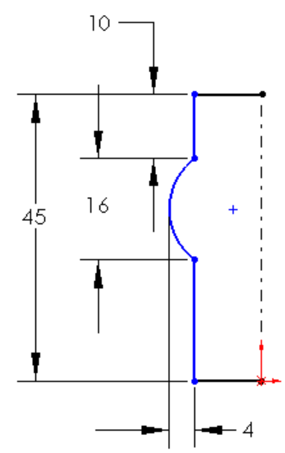


7 Finished dimension.
Change the **Value** to **4mm**.



8 Vertical dimensions.
Right-click empty space and select **More Dimensions** and **Vertical**. 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.

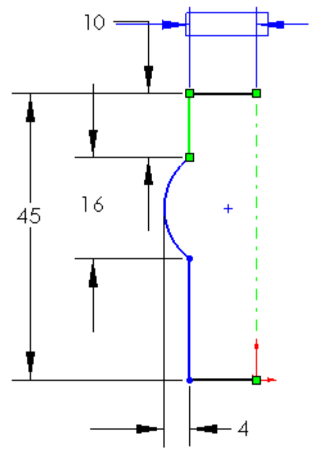
Note

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.

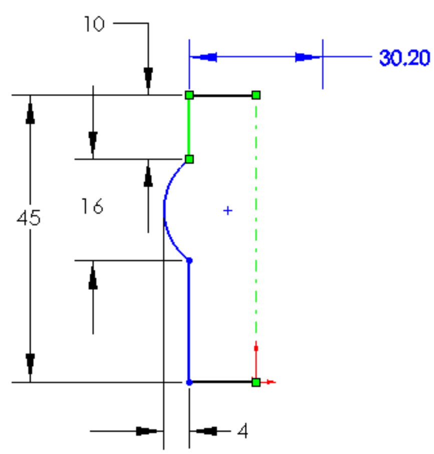
9 Dimension to centerline.
Dimension between the centerline and the outer vertical edge to create a horizontal linear dimension.

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.

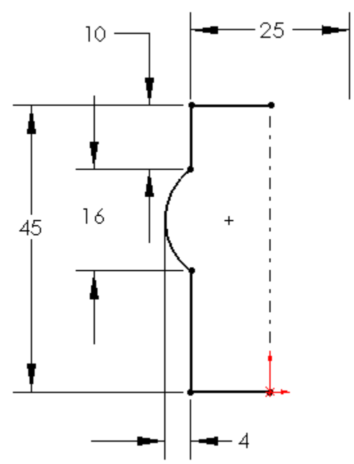


10 Move the cursor.
Move the cursor to the right of the centerline. The preview changes to a diameter dimension.



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. Click the dimension, and click the **Leaders** tab of the **Dimension** PropertyManager. Click the **Diameter** button 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 enables 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, on the **Features** toolbar, choose **Revolved Boss/Base** .

---

12 Make the feature.
Click **Boss/Base, Revolve...** from the **Insert** menu.

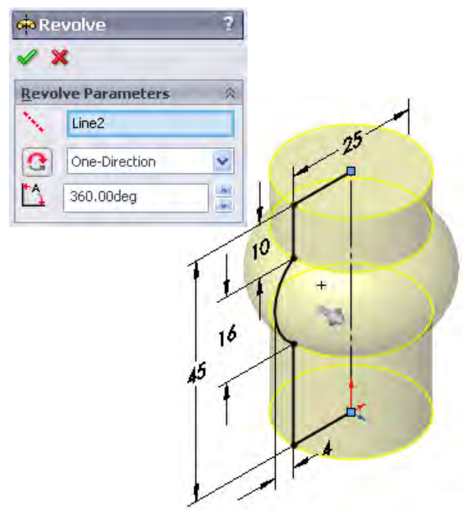
A message will appear indicating that the sketch is an open contour and asking if you want to close the contour automatically. Click **Yes**.

The PropertyManager appears with these default end conditions:

One Direction

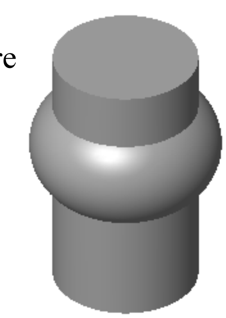
Angle 360°

Accept these defaults by clicking **OK**.



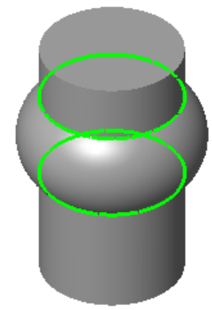
13 Finished feature.
The solid revolved feature is created as the first feature of the part.

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's true size and shape.

Introducing: Sketch Fillet

Sketch Fillets can be used to trim and add tangent arcs in a single step. If the corner has been trimmed, select the vertex point to add the fillet.

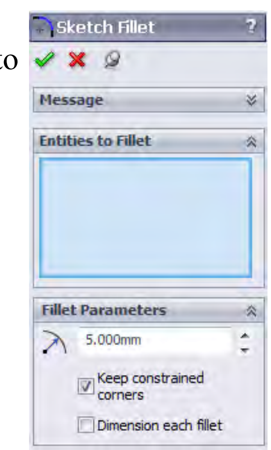
Where to Find It

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

---

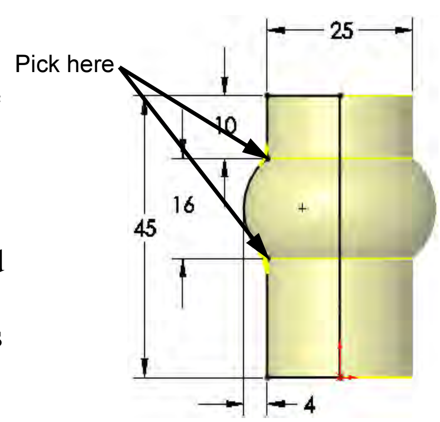
16 Fillet settings.

Select the **Sketch Fillet** tool and set the value to **5mm**. Make sure the **Keep constrained corners** option is checked.



17 Selections.

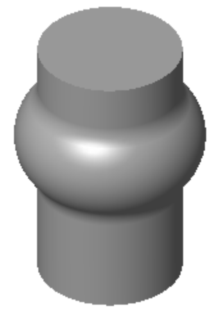
Select both endpoints of the arc, as indicated. When each is selected, the fillet will appear. The dimension drives both but only appears once, at the first selection. Click **OK**.



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.

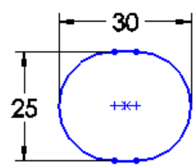
18 Rebuild the model.

To cause the changes to take effect click the **Rebuild** tool.

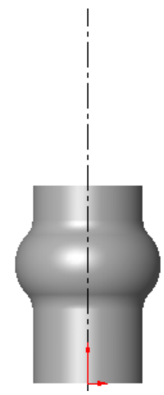


Building the Rim

The Rim of the Handwheel is another revolved feature. It too is revolved 360°. The profile of the Rim is a slot shape.



The Rim will be created as a separate solid body, not merged to the Hub.

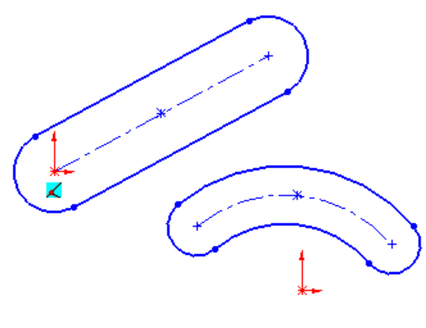


19 Sketch.

Create a new sketch on the Right plane. Orient the model in the same direction.

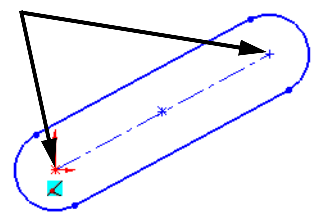
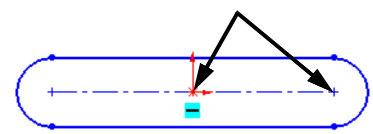
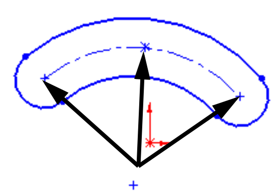
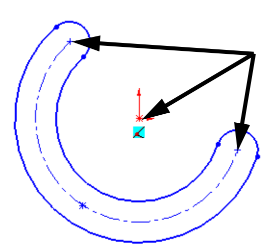
Slots

Straight and arc **Slots** are common shapes based on lines and arcs. The slot is a single entity which is composed of lines, arcs, construction geometry and points.



Introducing: Slots

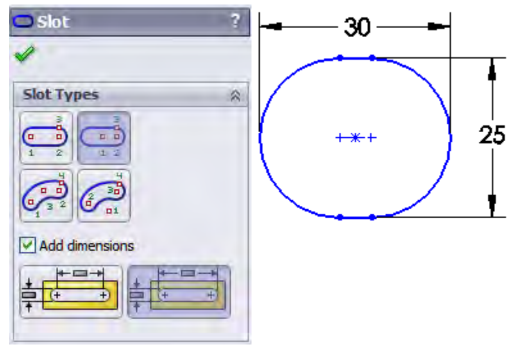
The **Slot** tool is used to create straight and arc slot shapes based on different criteria. There are two types based on lines and two types based on arcs. All slot types have the option to create dimensions with the geometry. The following types are available:

Slot Type	Resulting Geometry
Straight Slot	The **Straight Slot** is created by locating the centerpoints of the arcs and then dragging outwards to create the width. 
Centerpoint Straight Slot	The **Centerpoint Straight Slot** is created by locating the geometric center, one of the arc centerpoints and then dragging outwards to create the width. 
3 Point Arc Slot	The **3 Point Arc Slot** is created like a **3 Point Arc** (see *Introducing: 3 Point Arc* on page 192) and then dragging outwards to create the width. 
Centerpoint Arc Slot	The **Centerpoint Arc Slot** is created like a **Centerpoint Arc** (see *Sketch Geometry* on page 33) and then dragging outwards to create the width. 

Where to Find It

- From the **Tools** menu, choose **Sketch Entities, Straight Slot**, **Centerpoint Straight Slot**, **3 Point Arc Slot** or **Centerpoint Arc Slot**.
- Or, on the **Sketch** toolbar, choose **Straight Slot**, **Centerpoint Straight Slot**, **3 Point Arc Slot** or **Centerpoint Arc Slot**.

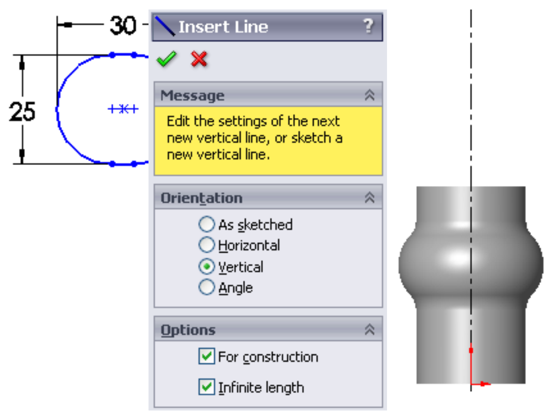
20 Centerpoint Straight Slot.
Click **Centerpoint Straight Slot**. Click **Add dimensions** and **Overall Length**. Click the location of the centerpoint and a location horizontally to the right. Drag the width and click **OK**.



Tip

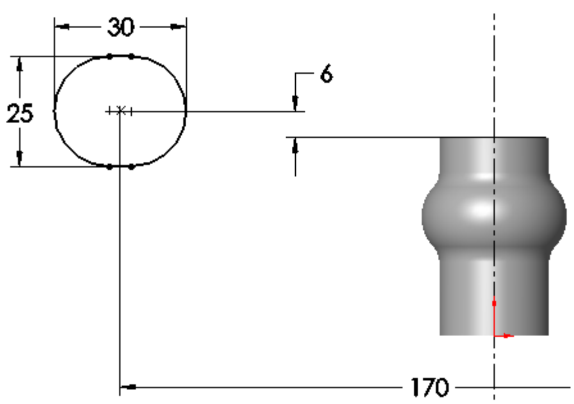
The dimensions are added automatically if the **Add dimensions** option is clicked.

21 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.



22 Add dimensions.
Add dimensions from the centerline to the point in the slot and the arc center to the Hub edge.

The sketch is now fully defined.



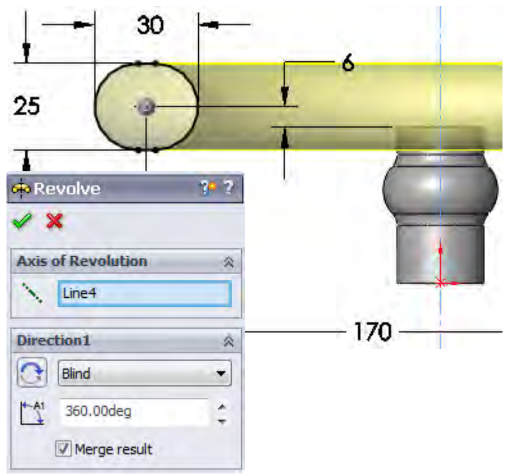
Potential Ambiguity

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.

23 Completed feature.

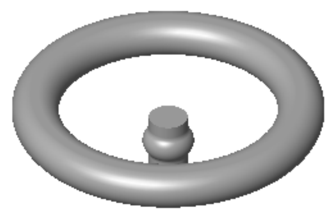
Select the infinite vertical centerline. From the **Insert** menu, choose **Boss/Base, Revolve...**. Use an angle of **360°**.

Rename the feature to Rim.



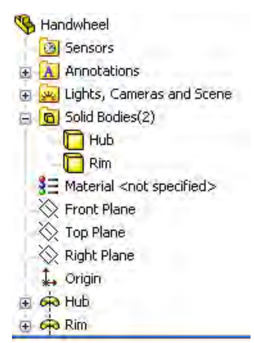
Multibody Solids

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.

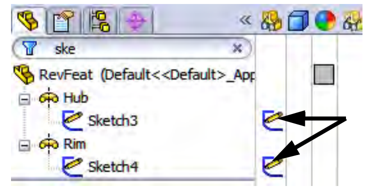


Building the Spoke

The Spoke feature is created using a **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.

24 Show using the display pane.



Use the FeatureManager Search box to search by the starting letters of a name or some portion of the name.

Type ske into the FeatureManager Design Tree filter to show the sketches of the Hub and Rim.

Click » to expand the Display Pane. Click on the sketch icon for the Hub to show it. Repeat for the Rim.

Click the "**x**" to clear the FeatureManager Design Tree filter and close the pane.

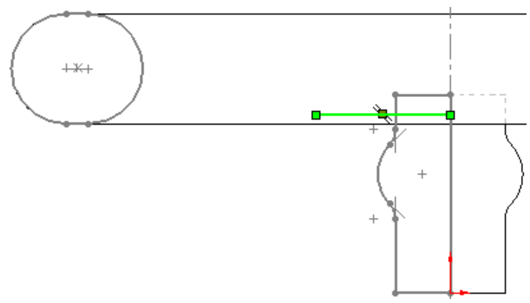
25 Setup.

Setup for sketching:

- Create a new sketch using the Right plane.
- Change the display to **Hidden Lines Visible**.

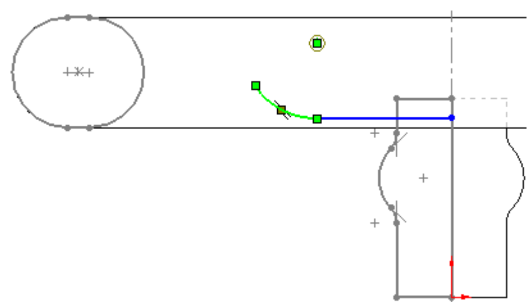
26 Sketch line.

Sketch a horizontal **Line** running from the centerline inside the Hub boundaries.



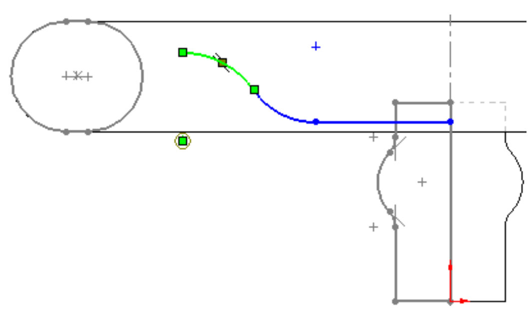
27 Tangent arc.

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.



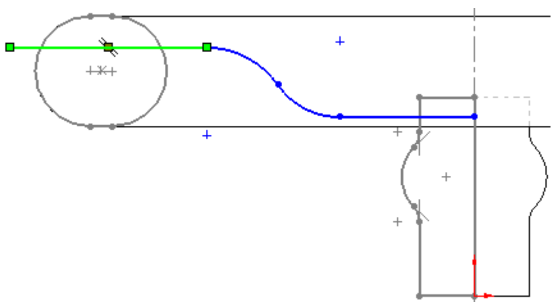
28 Connecting tangent arc.

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.



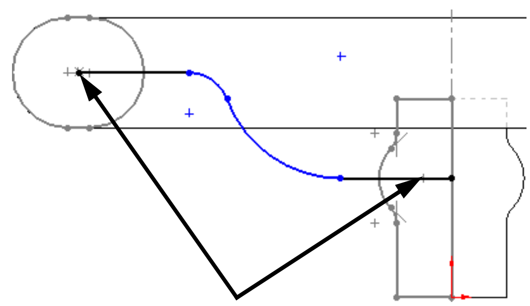
Tip When the vertical inference line coincides with the arc's center, the tangent of the arc is horizontal.

29 Horizontal line.
Sketch a final **Line**. It is horizontal, with its length to be determined by dimensioning.



30 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.

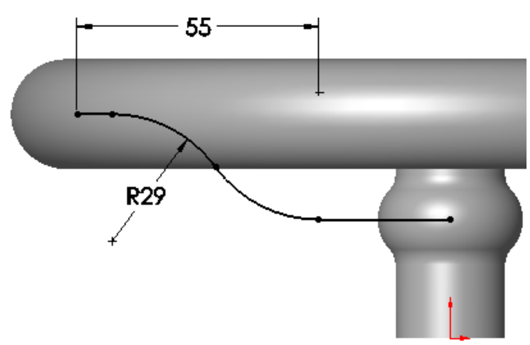


31 Return to a shaded display.
Click **Shaded** and hide the Hub and Rim sketches.

Completing the Path and Profile Sketches

The geometry sketched will act as the path for the profile sketch.

32 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 creating the dimensions.



33 Exit sketch.
Right-click in the sketch and choose **Exit Sketch** to close the sketch without using it in a feature.

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!

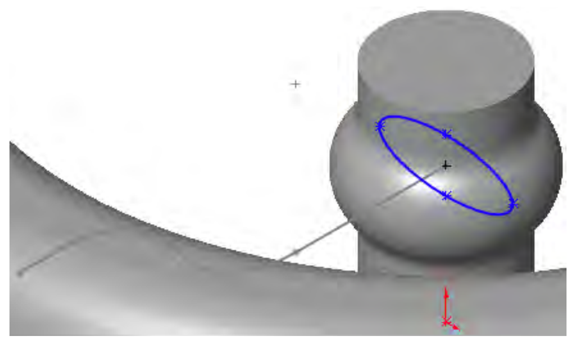
To fully define an ellipse you must dimension or otherwise constrain the lengths of the major and minor axes. You must *also* constrain the orientation of one of the two axes. One way to do this is with a **Horizontal** relation between the ellipse center and the end of the major axis.

Where to Find It

- Click **Tools, Sketch Entities, Ellipse**.
- Or, click **Ellipse** on the Sketch Tools toolbar.

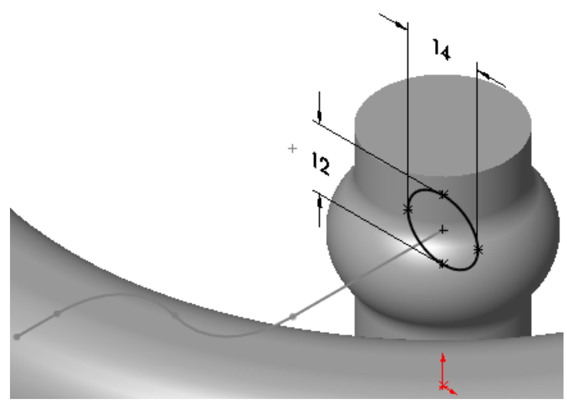
---

34 Ellipse.
Create a new sketch on the `Front` plane. Click **Ellipse** and position the centerpoint at the end of the line. Move away from center and position the major and minor axes with additional clicks.



35 Relations and dimensions.
Add relations to make the centerpoint and one of the major axis points **Horizontal**. Add the dimensions as shown.

Exit the sketch.



Introducing: Sweep

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

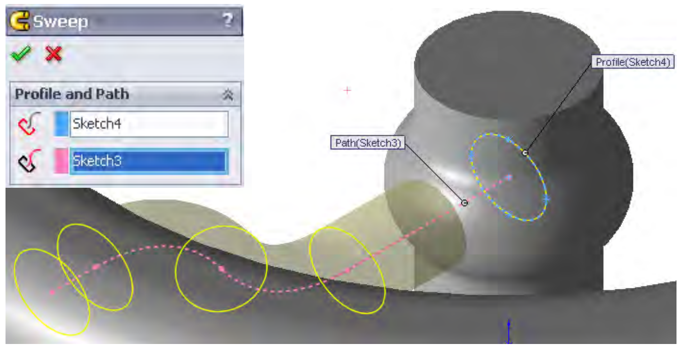
- Click **Swept Boss/Base** on the Features toolbar.
- Or, click **Insert, Base/Boss, Sweep**.

Note

The **Sweep** command is covered in depth in the *Advanced Part Modeling* course.

36 Sweep.

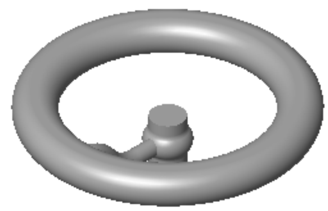
Click the **Swept Boss/Base** icon and select the closed contour sketch as the **Profile** and the open contour sketch as the **Path**.



Click **OK**.

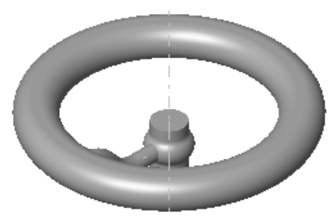
37 Results.

Name the new feature Spoke. The Solid Bodies(2) folder disappears. This indicates that the two solid bodies have merged into one.



38 Temporary axes.

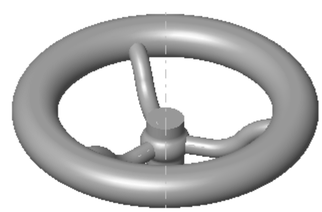
Display the temporary axes using **View, Temporary Axes**.



39 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 enables 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.

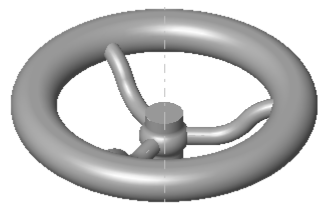
The same result can be obtained using the middle mouse button rotation. Select the temporary axis using the middle mouse button, and drag with the middle mouse button.

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.

---

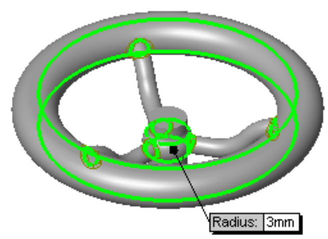
40 Rotate.
Rotate about the axis by dragging the mouse. Switch axes by simply clicking another axis or other acceptable choice.



Turn off the temporary axes.

41 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.

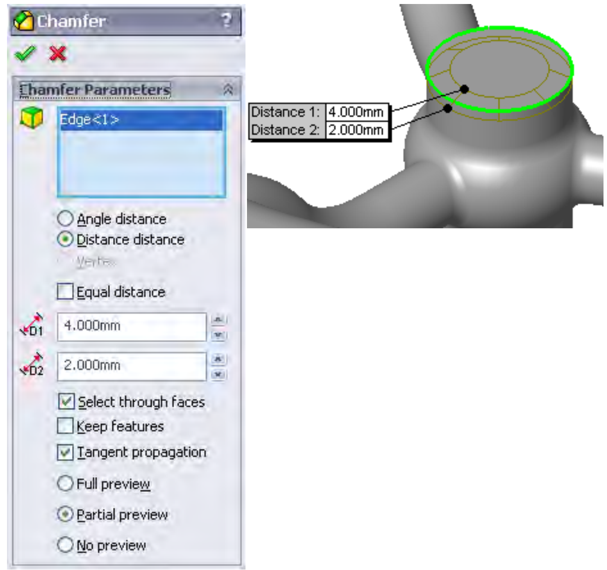
Note

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.

42 Chamfer.
Add a **Chamfer** feature using the top edge of the Hub feature. Set the distances using the values shown at right.



RealView Graphics

If you have an NVIDIA, ATI or 3DLabs graphics accelerator, you may be able to use the **RealView Graphics** option. It provides high-quality, real time material shaders when available.

Where to Find It

- Click **RealView Graphics** on the View toolbar and click the RealView tab of the Task Pane.
- Or, click **View, Display, RealView Graphics**.

Note

If you do not have RealView Graphics, skip to step **47** on page 209.

Tip

If **RealView Graphics** are not available, the icon will be grayed out and the RealView tab shown below will not be available.

RealView On

RealView Off

Appearances, Scenes and Decals

The **Appearances, Scenes and Decals** tab of the Task Pane contains three main folders: Appearances(color), Scenes and Decals.

---

43 RealView on.

Click **RealView** to toggle it on.

44 Appearances and scenes.

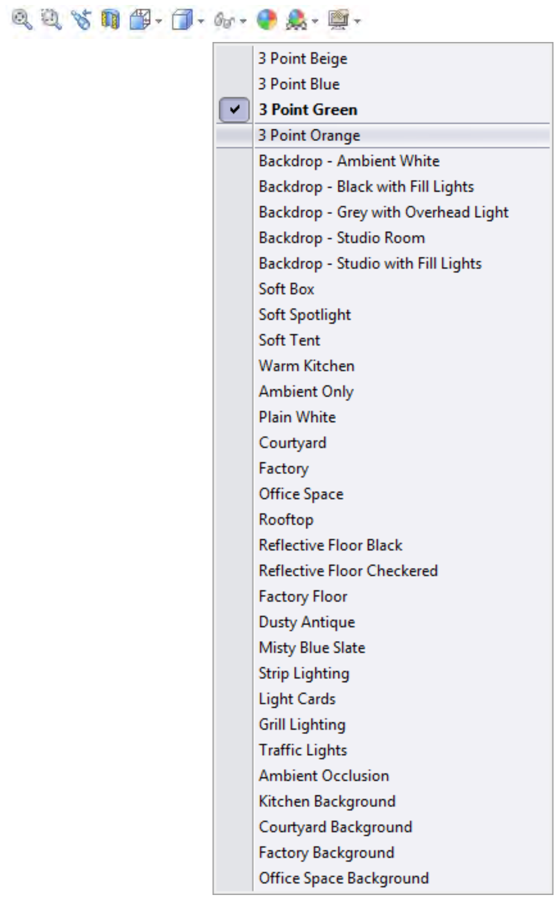
From the **Appearances**, **Painted**, **Powder Coat** folder, drag and drop **aluminum powdercoat** into the graphics window.

From the **Scenes**, **Basic Scenes** folder, drag and drop **Backdrop - Black with Fill Lights** into the graphics window.

Tip

The **Apply Scene** flyout tool on the **Heads-up View** toolbar allows you to select and apply a scene from the list.

Another option is to click the icon to rotate through the list one at a time.



Appearances

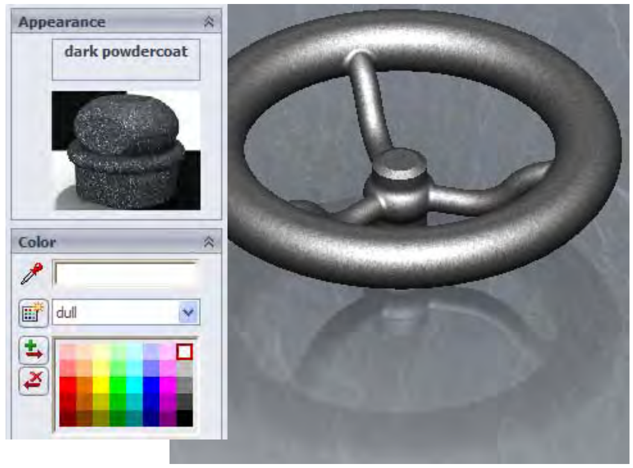
Colors and textures are applied using **Appearances**. This menu has tabs for **Color/Image** and **Mapping**.

- **Color** is used to apply a color to the texture added from the **Appearance** folder.
- **Mapping** is used to change the mapping style of the texture added from the **Appearance** folder.

Where to Find It

Right-click the top level part, choose **Appearances** and the part name.

45 Color.
Right-click the top level part and choose **Appearances** and the part name. Lighten the color using a dull color swatch and light grey or white. Click **OK**.



Note

Applying an appearance does not apply a material to the part. For applying materials, see *Edit Material* on page 209.

46 RealView off.
Click **RealView** to toggle it off.

47 Save the part and close it.

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 **SimulationXpress**. The material can vary by configuration. For more information on configurations, see *Lesson 10: Configurations*.

Tip

Part templates (`*.prtdot`) can include a predefined material.

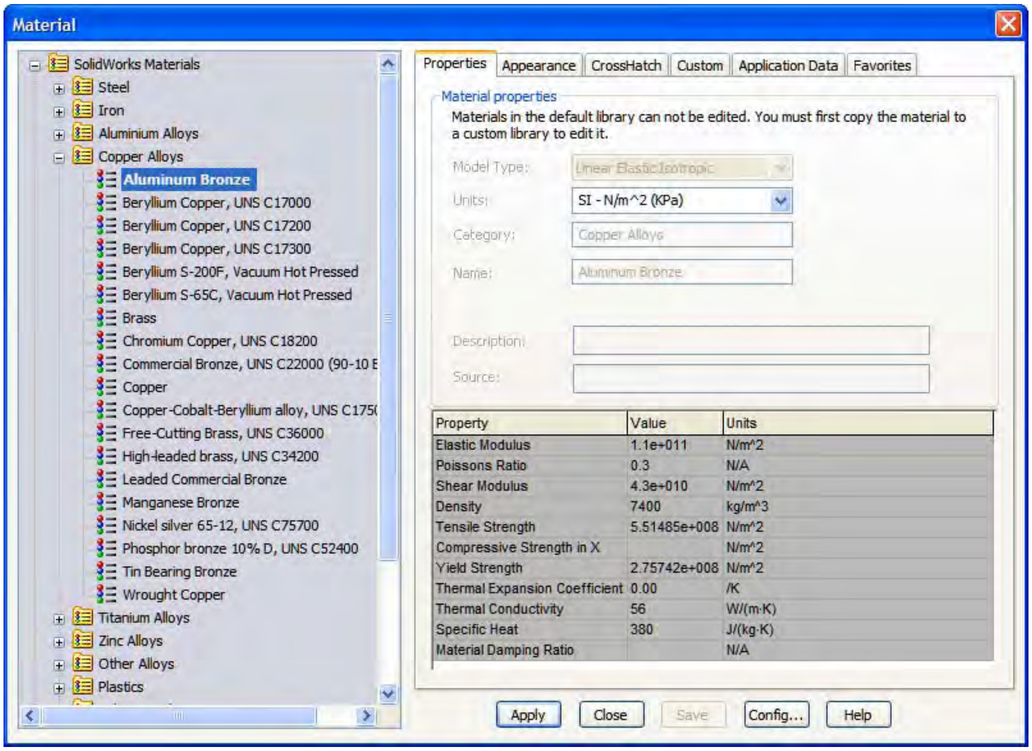
Where to Find It

- Click **Edit Material** on the Standard toolbar.
- Or, right-click the Material icon and click **Edit Material**.

1 Open `HW_Analysis`.
Open the existing part `HW_Analysis`. This part has additional features needed for use in the analysis section of this lesson.

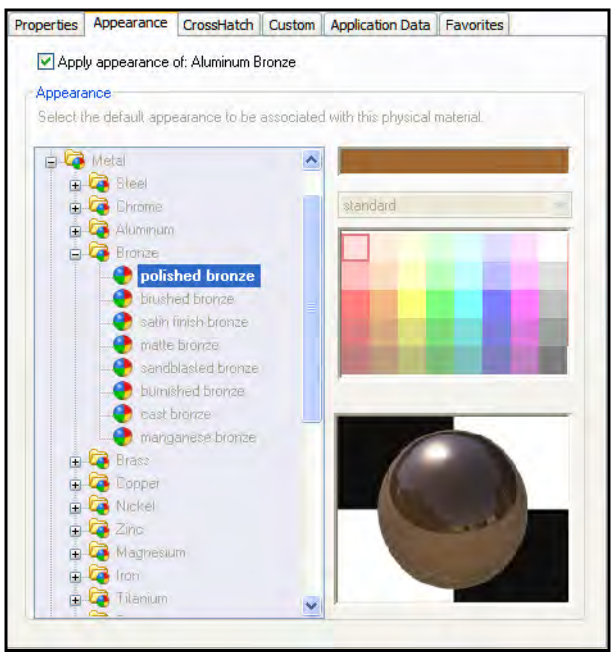
2 Materials.

Click the **Edit Material** icon and select **Copper Alloys**, **Aluminum Bronze**.

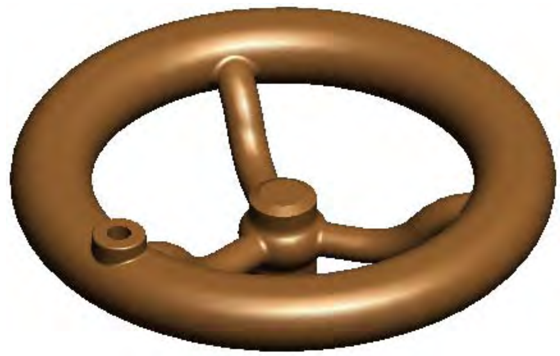


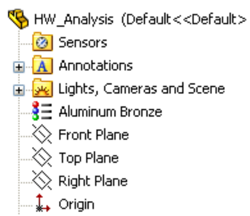
Note

The **Properties, Appearance** and C**rossHatch** are those assigned by the selected material.



3 Color.
Click **Apply** and **Close**. A change in material changes the color of the part. 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.

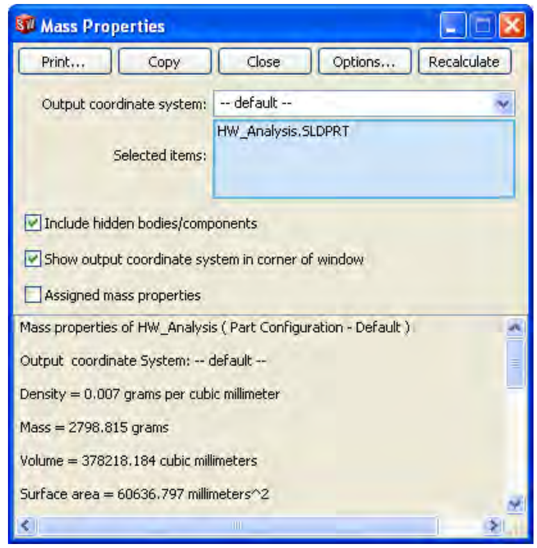
Where to Find It

- On the Tools toolbar, click the **Mass Properties** tool.
- From the **Tools** menu choose **Mass Properties...**.

4 Mass properties.

Select the **Mass Properties...** option from the **Tools** menu. The **Density** of Aluminum Bronze is used.

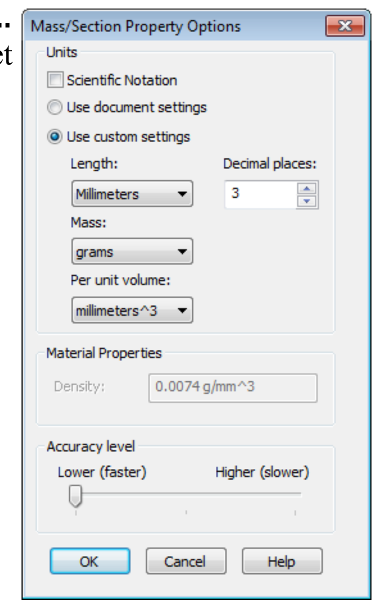
The results of the calculations are displayed in the dialog box.



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).

To change the settings, click the **Options...** button, click **Use custom settings**, 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**.



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.

File Properties

File properties are details about Windows based files that help identify it – for example, a descriptive title, the author name, the subject, and keywords that identify topics or other important information in the file. Document properties can be used to display information about a file or to help organize files so that they can be found easily. You can search for documents based on document properties.

There are file properties unique to SolidWorks that are more suited to engineering than the default properties. Additional properties can be added based on the user's needs.

Metadata

File properties and attributes are sometimes referred to as Metadata.

Classes of File Properties

File properties can be grouped into several classes.

- **Automatic**
  Automatic properties are maintained by the application that created the property. These include properties such as the date the file was created, last modified and file size.
- **Preset**
  Preset properties already exist, but the user must fill in the text value. The preset file properties used in SolidWorks are stored in the file `Property.txt`. This file may be edited to add or remove preset properties.
- **Custom**
  Custom properties are defined by the user and apply to the entire document.
- **Configuration specific**
  Configuration specific properties apply only to a specific configuration.
- **SolidWorks custom properties**
  There are several custom properties that can be automatically updated by SolidWorks. These include the part's mass and material.

Tip

Preset properties can accept text, date, yes/no and numerical data.

Where to Find It

- Click **File, Properties**.

Creating File Properties

File properties can be created directly in the file, or they can be created by other procedures.

- **Direct method**
  File properties are added directly to the file by the user.
- **Design tables**
  Design tables can create custom properties using a column header **$PRP@property** where **property** is the name of the property to be created and populated with the information created in the design table.
- **SolidWorks Workgroup PDM**
  SolidWorks Workgroup PDM will add several custom properties to files checked into the vault. These include: number, status, description, project and revision. SolidWorks Workgroup PDM can also be configured to add additional properties defined by the Vault Administrator.

Uses of File Properties

File properties can be used for several operations.

- **Parts, assemblies and drawings**
  File properties can be used to create parametric notes. Annotations linked to file properties will update as the properties change.
- **Assemblies**
  Advanced Selection and **Advanced Show/Hide** can select components based on specific file properties. Specific procedures are found in the training course *Assembly Modeling*.
- **Drawings**
  File properties can be used to fill in data in the title block, BOM, revision blocks and annotations. Specific procedures are found in the training course *SolidWorks Drawings*.

All SolidWorks documents have the following system-defined properties:

Property Name	Value
SW-Author	Author field in Summary Information dialog box
SW-Comments	Comments field in Summary Information
SW-Configuration Name	Configuration name in ConfigurationManager of part or assembly
SW-Created Date	Created field in Summary Information
SW-File Name	Document name without extension
SW-Folder Name	Document folder with a backslash at the end
SW-Keywords	Keywords field in Summary Information
SW-Last Saved By	Last Saved By field in Summary Information
SW-Last Saved Date	Last Saved field in Summary Information
SW-Long Date	Current date in long format (Monday, January 1, 2005)
SW-Short Date	Current date in short format (1/1/2005)
SW-Subject	Subject field in Summary Information
SW-Title	Title field in Summary Information

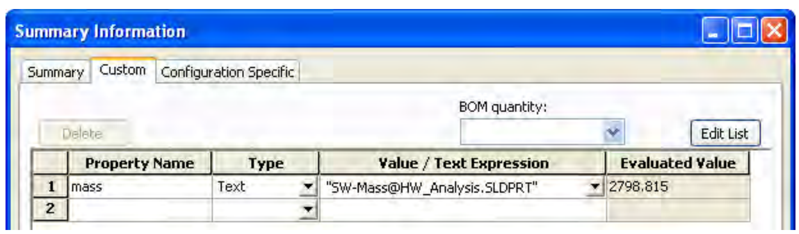
Additionally, drawings have the following system-defined properties:

Property Name	Value
SW-Current Sheet	Sheet number of the active sheet
SW-Sheet Format Size	Sheet size of the active sheet format
SW-Sheet Name	Name of the active sheet
SW-Sheet Scale	Scale of the active sheet
SW-Template Size	Template size of the drawing template
SW-Total Sheets	Total number of sheets in the active drawing document

Prefixes for custom properties linked in notes are used as follows:

Prefix	Evaluated From
$PRP	Current document
$PRPSHEET	Model in the view specified in Sheet Properties For sheet and format notes, the first view in the FeatureManager design tree is used
$PRPVIEW	Model in the drawing view to which the note belongs
$PRPMODEL	Component to which the annotation is attached

5 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** drop-down list. A SolidWorks special property and **Evaluated Value** are created.



Note

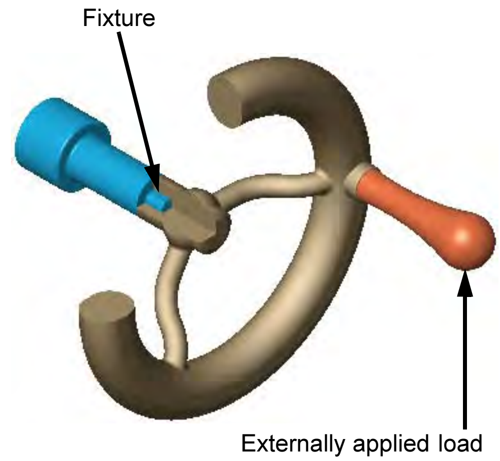
The **Configuration Specific** tab can also be used. This would allow the property to vary by configuration. Configurations will be discussed in *Lesson 10: Configurations*.

SolidWorks Simulation-Xpress

SolidWorks SimulationXpress 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. SolidWorks SimulationXpress is a subset of the SolidWorks Simulation product.

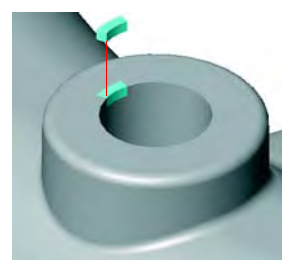
Overview

SolidWorks SimulationXpress 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: *fixtures*, *loads* and *materials*. 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 *fixture* - the hub is restrained so it doesn't move. Fixtures are sometime called *constraints*. 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, SolidWorks SimulationXpress 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 SolidWorks Simulation-Xpress

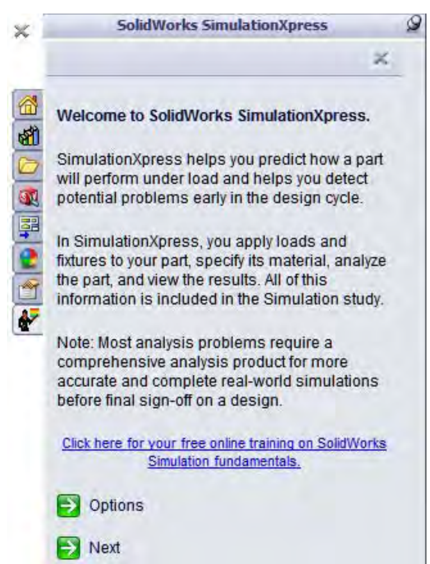
SolidWorks SimulationXpress walks you through the steps of analysis, from **Options** to **Optimize**. The steps are:

- **Options**
  Setup the type of units that are commonly used for materials, loads and results.
- **Fixtures**
  Select faces of the part that remain in place (fixed) during the analysis.
- **Loads**
  Add external loads such as forces and pressures to induce stress and to deform the part.
- **Material**
  Choose a material for the part from the standard library or input your own.
- **Run**
  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*.
- **Optimize**
  Optimize a result quantity using a selected dimension.

Where to Find It

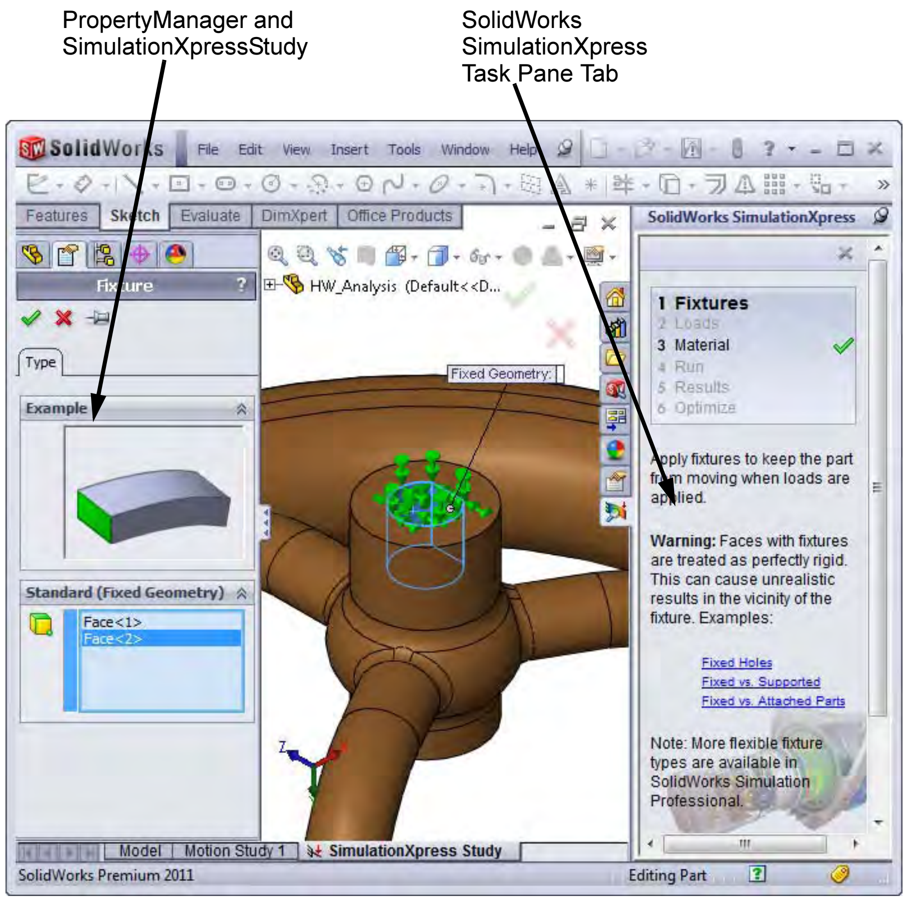
- From the **Tools** menu, select **SimulationXpress...**.
- Or, on the Tools toolbar, click **SimulationXpress** .

1 Start SimulationXpress.
Click **Tools, SimulationXpress...**.
The analysis wizard appears in the Task Pane.



The SimulationXpress Interface

The **SolidWorks SimulationXpress** interface begins in the Task Pane, where the checklist of sequential tasks appears under the SolidWorks SimulationXpress tab. Selection options and the SimulationXpressStudy tree appear in the PropertyManager.



Options

The **Options** dialog contains settings for the **System of units** and **Results location**.

2 **Click Options....**
Set the units to **SI** and use the default **Results location**.

Click **OK** and click **Next**.

Phase 1: Fixtures

Fixtures 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. As you complete each phase in the wizard, a green check mark is added to the list.

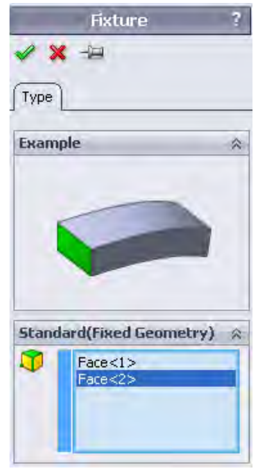
3 **Fixtures page.**
Click **Add a fixture**.

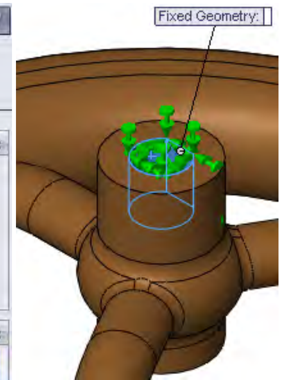
Tip Click on the blue hyperlinks (such as Fixed Holes) for examples.

4 Face selection.
Select the cylindrical face and the flat face that form the D-shaped hole.

Click **OK**.

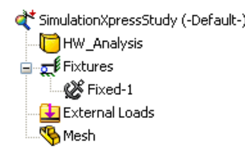
Click **Next**.





Simulation Study

A **SimulationXpress Study** is being constructed as the wizard steps are completed. The FeatureManager Design Tree is split and the lower portion contains the SimulationXpressStudy tree.



Eventually it will include fixtures, loads, mesh and the results of the analysis.

Phase 2: Loads

The **Loads** page 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.

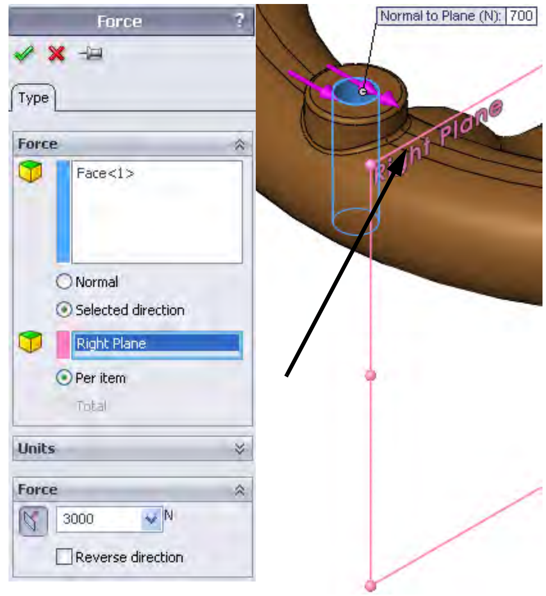
Note

The specified force value is applied to *each* face. For example, if you select 3 faces and specify a 50 lb. force, SimulationXpress applies a total force of 150 lbs. (50 lbs. on each face).

5 Loads page.
In this example, we will use a **Force** type load. Click **Add a force**.

6 Select the face.
Select the cylindrical face as shown. Click **Selected direction** and click the Right plane.

Set the **Force** to **3000** and click **OK**.



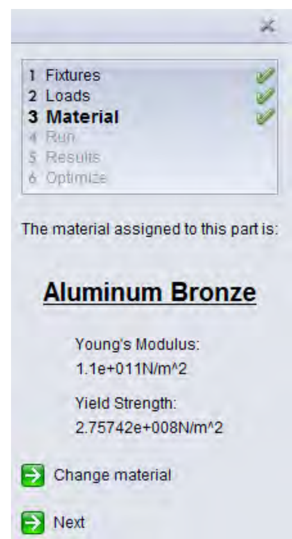
Phase 3: Material

The next phase is selecting the **Material**. You can choose from libraries of standard materials or add your own.

7 Material page.
The current material, selected within SolidWorks, should be **Aluminum Bronze** from the **Copper Alloys** list.

To change the material, select it from the list. This is the same list that appears when using **Edit Material**.

Click **Next** to keep Aluminum Bronze as the material.



Phase 4: Run

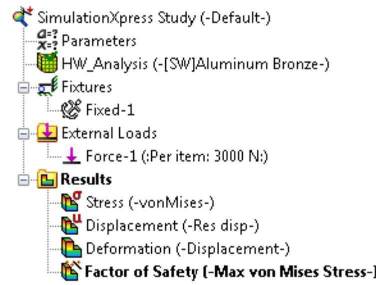
SimulationXpress prepares the model for analysis, creates the mesh and calculates displacements, strains, and stresses.

8 Run page.
The required information has been provided and the analyzer is ready.

Click **Run simulation**.

Phase 5: Results

The **Results** page is used to display the results of analysis. The full SimulationXpressStudy tree is shown in the split FeatureManager Design Tree. This includes all the input and output of the study.



Tip

Right-click a result feature (such as `Stress (-vonMises-)`) and choose **Show** to view it.

9 Deformation.
The first result displayed is the **Deformation**.

Click **Yes, continue** to see the next result.

The second result is the **Factor of Safety** (FOS) which compares the yield strength of the material to the actual stresses.

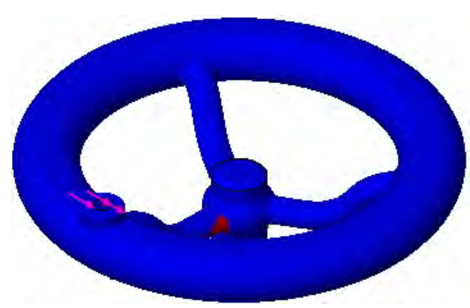
Factor of Safety

SimulationXpress 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. SimulationXpress 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.

10 Factor of safety.
The **FOS** has areas that are less than **1**. This indicates that areas of the part are overstressed and will fail.



Red areas indicate where the factor of safety is less than one.

Click **Done viewing results**.

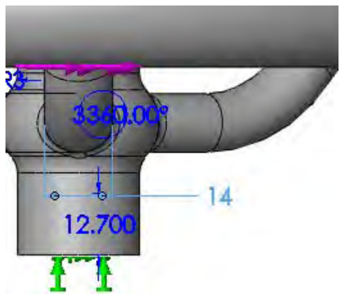
Click **6 Optimize**.

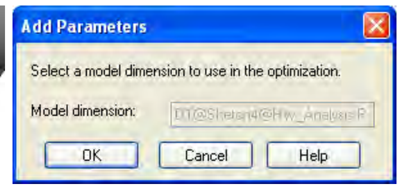
Phase 6: Optimize

The **Optimize** tab can be used to bring **Factor of Safety**, **Max Stress** or **Maximum Displacement** values to acceptable levels by iterating the value of a dimension. The optimization is performed within set numeric boundaries with the above constraints.

11 Optimize the model.
Click **Yes** for **Would you like to optimize your model?** and **Next.**

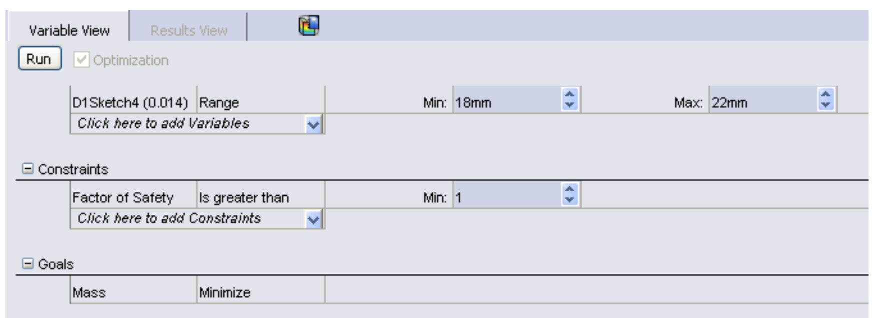
12 Value to change.
Select the **14mm** dimension (major axis of the ellipse) as the dimension to change. Click **OK**.





13 Variables and constraints.
Set the **Min** and **Max Variable** values to **18mm** and **22mm** as shown. For **Constraints**, select **Factor of Safety** and set it to a minimum of **1**.

Click **Run**.



Tip **Design Study Properties** can be used to set the results quality and output folder location.

14 Results.
After several iterations, the optimization is complete. Click the **Results view** tab. The resulting changes meet the FOS target with only a small increase in weight.

Variable View | Results View

	Initial	Optimal
D1Sketch4	14mm	20.2804mm
Factor of Safety	0.635995	1.017933
Mass	2.79881 kg	2.88251 kg

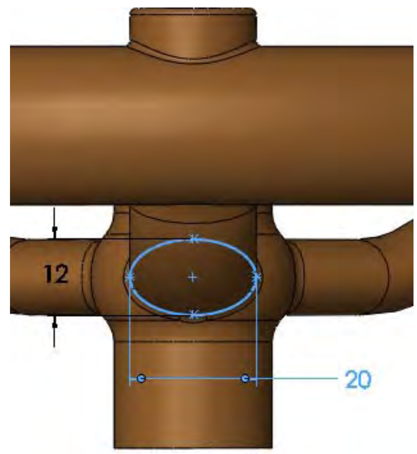
15 Optimization results.
Click the **Optimal Value** option and click **Next**. Click **4. Run** and **Run Simulation**.

Updating the Model

Changes performed in SolidWorks or during optimization are detected by SimulationXpress. Changes can be made to the model, materials, fixtures or loads. The existing analysis can be re-analyzed to show the newest results.

16 Save data.
Click **Close** in the SolidWorks SimulationXpress window and **Yes** to save the data.

17 Change model.
The optimization process changed the selected dimension value. Click the **Model** tab on the bottom of the graphics window and change that dimension to the rounded value **20mm** and rebuild.



18 Retrieve data.
Start SimulationXpress again and run the simulation again.

19 Save and close.
Click **Close** in the SolidWorks SimulationXpress window and **Yes** to save the data.

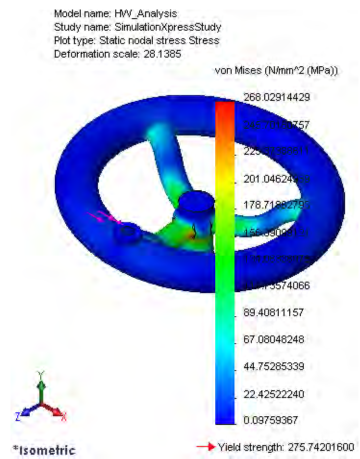
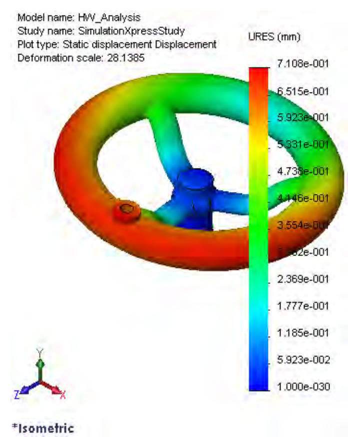
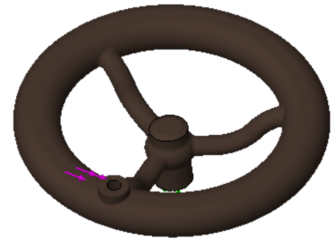
20 Save and close the part.

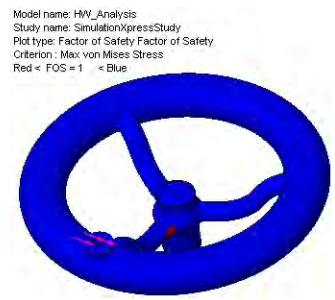
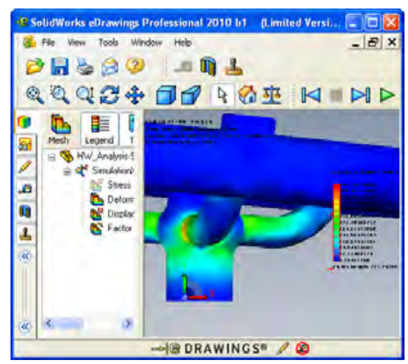
Results, Reports and eDrawings

The following are some examples of the different types of output that is available with an analysis. It includes results, reports and eDrawing files.

Note

Some of the displays are exaggerated by a deformation scale.

Type	Display
Stress (-vonMises-)	
Displacement (-Res disp-)	
Deformation (-Displacement-)	

<table>
<tr><th>Type</th><th>Display</th></tr>
<tr><td>Factor of Safety (-Max von Mises Stress-)</td><td>
</td></tr>
<tr><td>HTML Report</td><td>
<table>
<tr><th colspan="2">Mesh Information</th></tr>
<tr><td>Mesh Type:</td><td>Solid Mesh</td></tr>
<tr><td>Mesher Used:</td><td>Standard mesh</td></tr>
<tr><td>Automatic Transition:</td><td>Off</td></tr>
<tr><td>Smooth Surface:</td><td>On</td></tr>
<tr><td>Jacobian Check:</td><td>4 Points</td></tr>
<tr><td>Element Size:</td><td>7.2337 mm</td></tr>
<tr><td>Tolerance:</td><td>0.36169 mm</td></tr>
<tr><td>Quality:</td><td>High</td></tr>
<tr><td>Number of elements:</td><td>9894</td></tr>
<tr><td>Number of nodes:</td><td>16393</td></tr>
<tr><td>Time to complete mesh(hh;mm;ss):</td><td>00:00:06</td></tr>
<tr><td>Computer name:</td><td>TRN-FKOEHLER740</td></tr>
</table></td></tr>
<tr><td>eDrawings File</td><td>
</td></tr>
</table>

Exercise 24: Flange

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- *Revolved Features* on page 191.

Units: **millimeters**

Design Intent

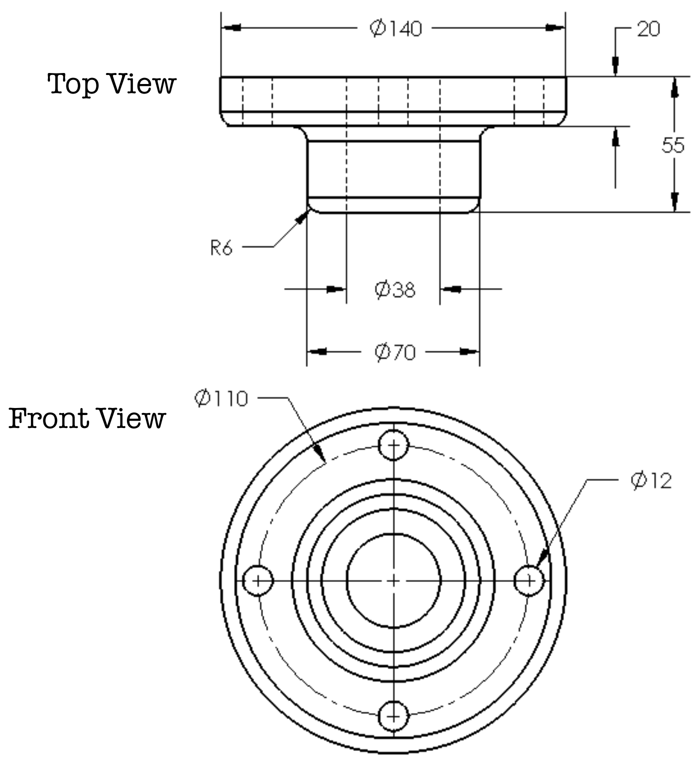
The design intent for this part is as follows:

1. Holes in the pattern are equally spaced.
2. Holes are equal diameter.
3. All fillets are equal and are **R6mm**.

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.



Exercise 25: Wheel

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- *Revolved Features* on page 191.

Units: **millimeters**

Design Intent

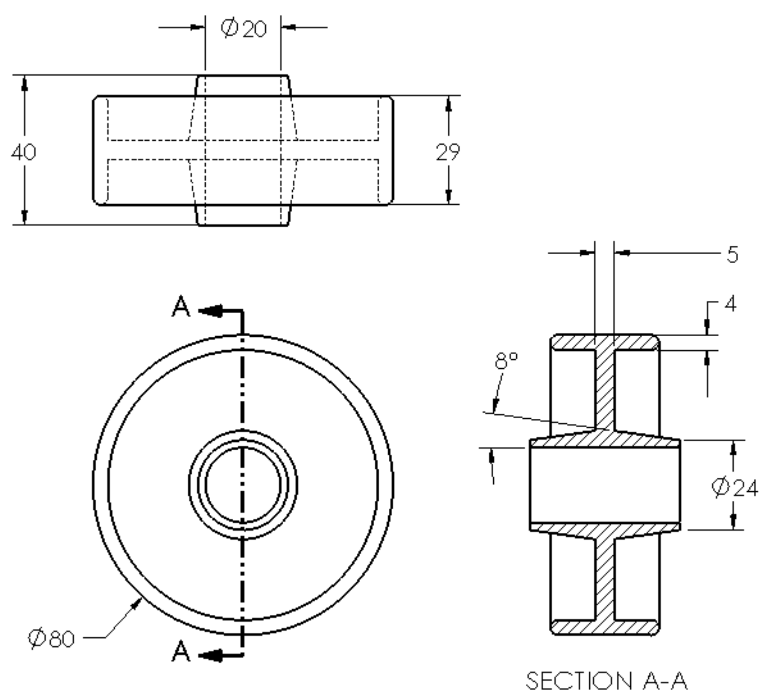
The design intent for this part is as follows:

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.



Optional: Text in a Sketch

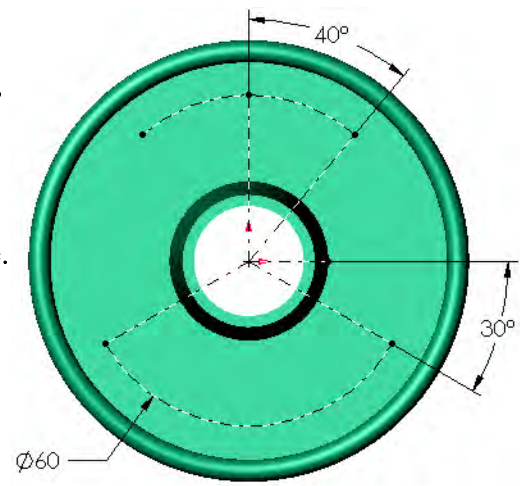
Text can be added to a sketch and extruded to form a cut or a boss. The text can be positioned freely, located using dimensions or geometric relations, or made to follow sketch geometry or model edges.

Where to Find It

- Click **Tools, Sketch Entities, Text...**.
- Or, on the Sketch toolbar, click **Text**.

---

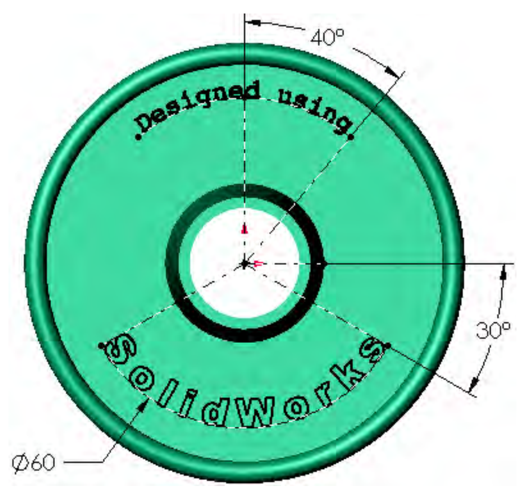
1 **Construction geometry.**
Sketch on the front face and add construction lines and arcs as shown.



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°**.

Note

Extruding text can be time consuming.

4 **Save the part and close it.**

Exercise 26: Guide

Create this part using the information and dimensions provided. This lab reinforces the following skills:

- *Introducing: Slots* on page 198.

Units: **millimeters**

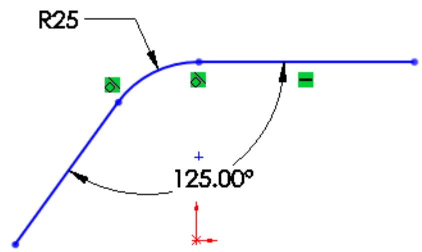
Procedure

Create a new mm part and name it `Guide`. Create the geometry as shown in the following steps.

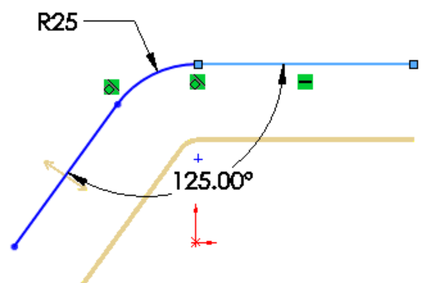
Note

These images show the sketch relations (**View, Sketch Relations**) for clarity.

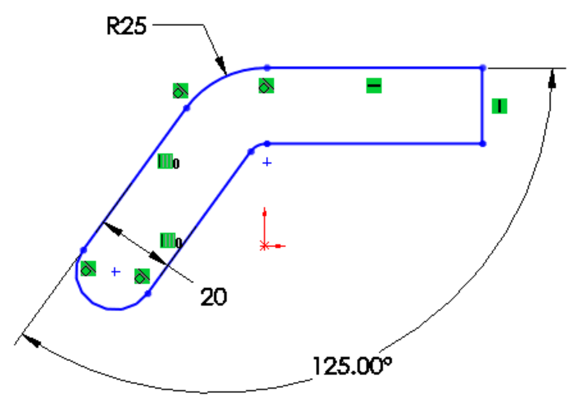
1 Lines and fillet.
Open a sketch on the `Front` plane. Create the sketch lines, a sketch fillet and an angular dimension as shown.



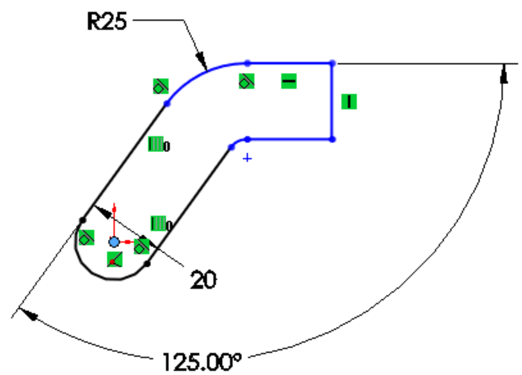
2 Offset.
Use offset entities to create the **20mm** offset as shown.



3 Close ends.
Close the ends using a tangent arc and a line as shown.

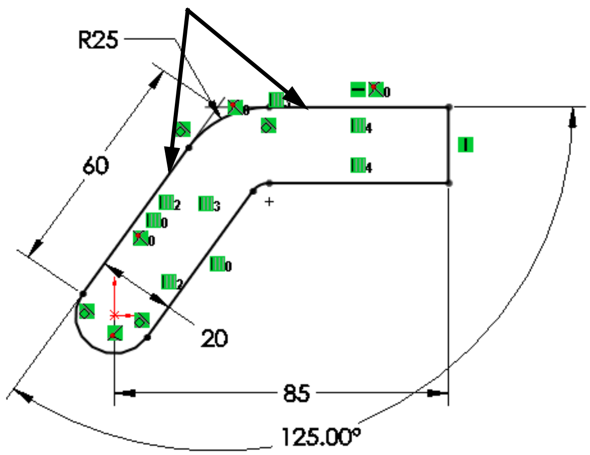


4 Drag to origin.
Drag the centerpoint of the arc to the Origin and drop it. This creates a Coincident relation.

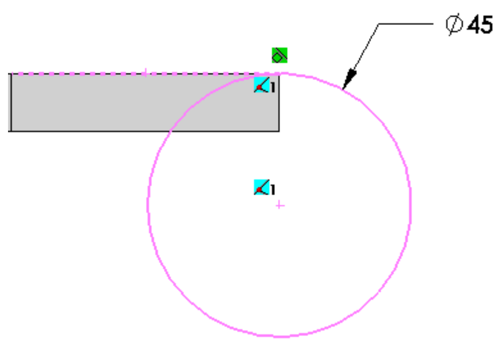


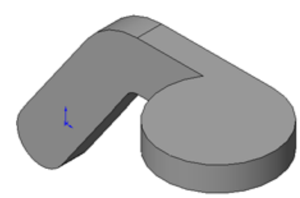
5 Fully defined.
Add a Virtual Sharp by selecting the two lines as shown and clicking the **Point** tool.
Complete the sketch by adding dimensions as shown.

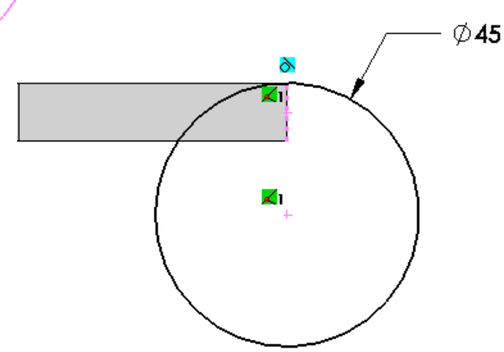
Extrude the sketch **10mm**.



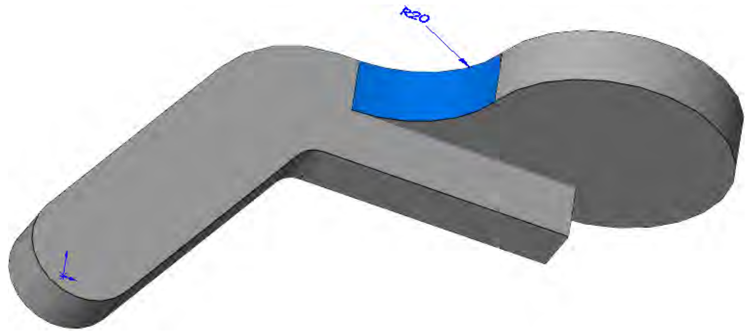
6 Circle and boss.
Add a circle to a new sketch on the top face of the model. Use **Tangent** and **Coincident** relations to relate the circle to the geometry. Fully define and extrude the sketch **10mm** as shown.



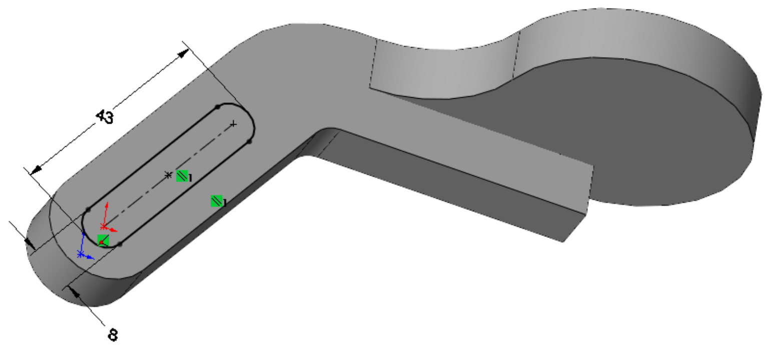




7 Fillet.
Add a fillet **R20mm** as shown.



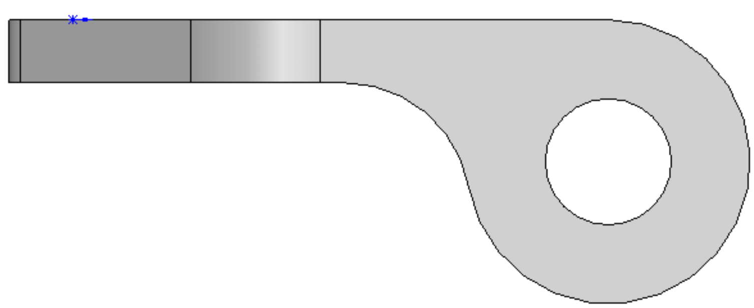
8 Slot.
Use **Straight Slot** with the options **Overall Length** and **Add Dimensions** to create the geometry shown below. Create a through all cut with the sketch geometry.



Tip

The slot sketch should be fully defined. It may require a **Parallel** relation.

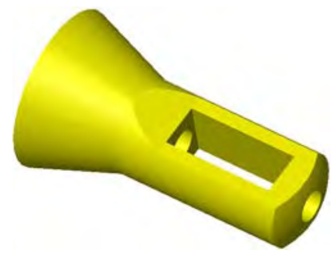
9 Hole.
Add a **20mm** through all hole to complete the part.



10 Save and close.

Exercise 27: Tool Post

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.



This lab uses the following skills:

- *Revolved Features* on page 191.

Units: **millimeters**

Design Intent

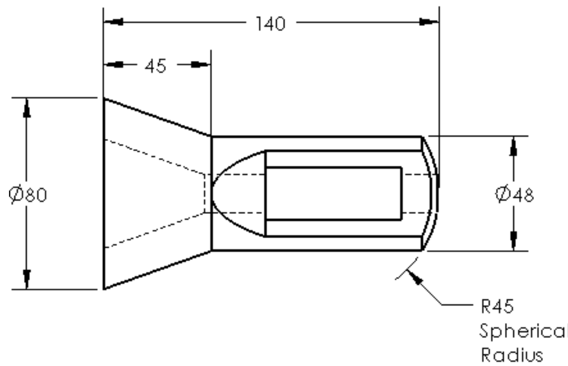
The design intent for this part is as follows:

1. Part is symmetrical.
2. Center hole is though all.

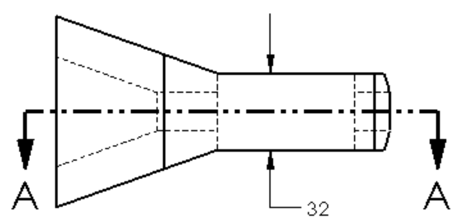
Dimensioned Views

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

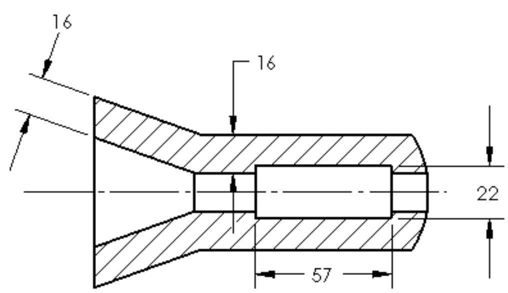
Top View



Front View



Section A-A



SECTION A-A

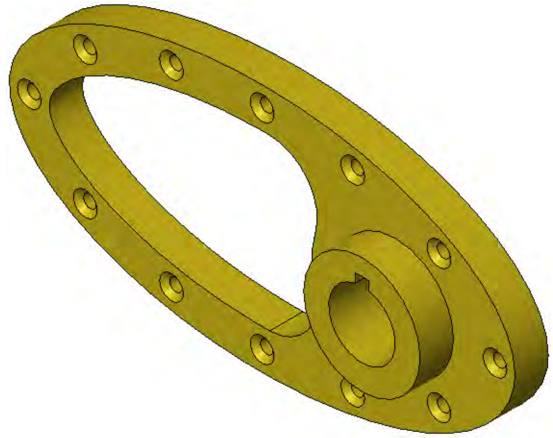
Exercise 28: Ellipse

Use an ellipse to create the part.

This lab reinforces the following skills:

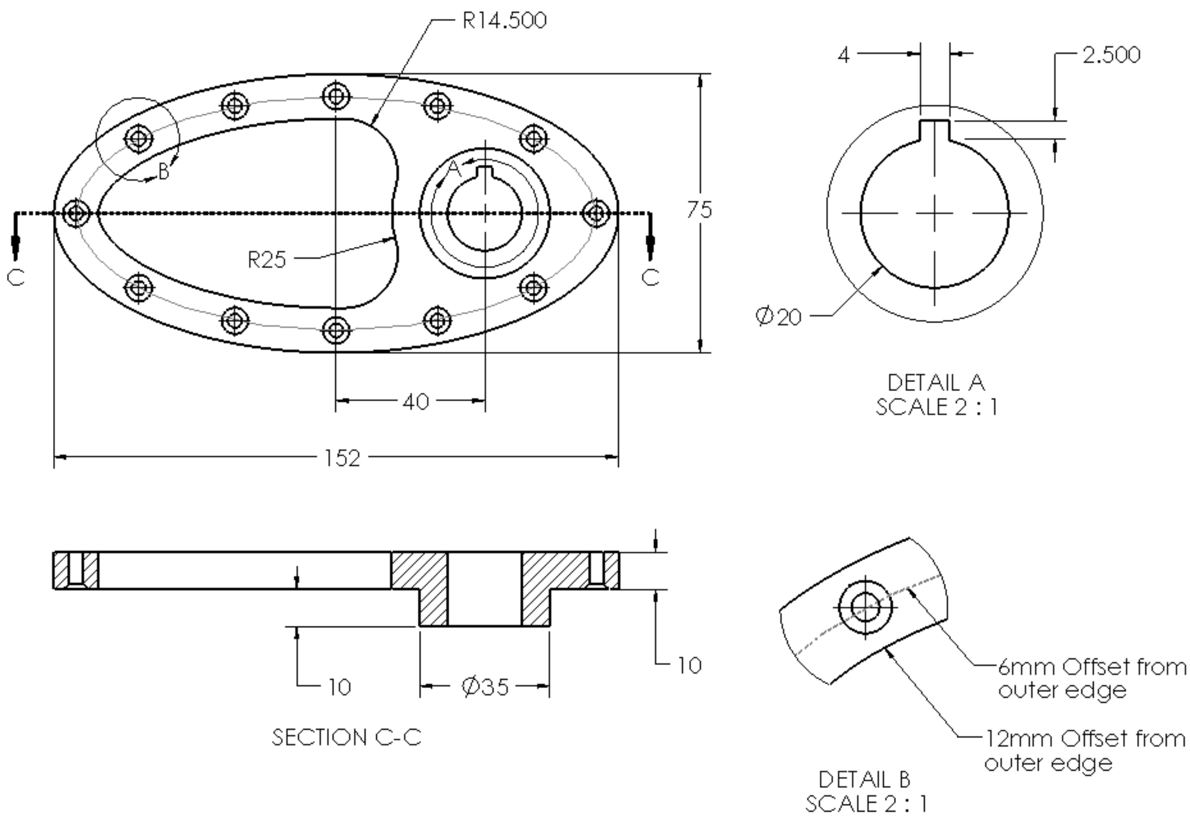
- *Introducing: Insert Ellipse* on page 203.

Units: **millimeters**



Procedure

Create a new mm part and name it `Offsets`. All **Countersink** holes are for an **M3 Flat Head Machine Screw**. Create the geometry as shown in the following steps.



Exercise 29: Sweeps

Create these three parts using swept features. These require a path and a section.

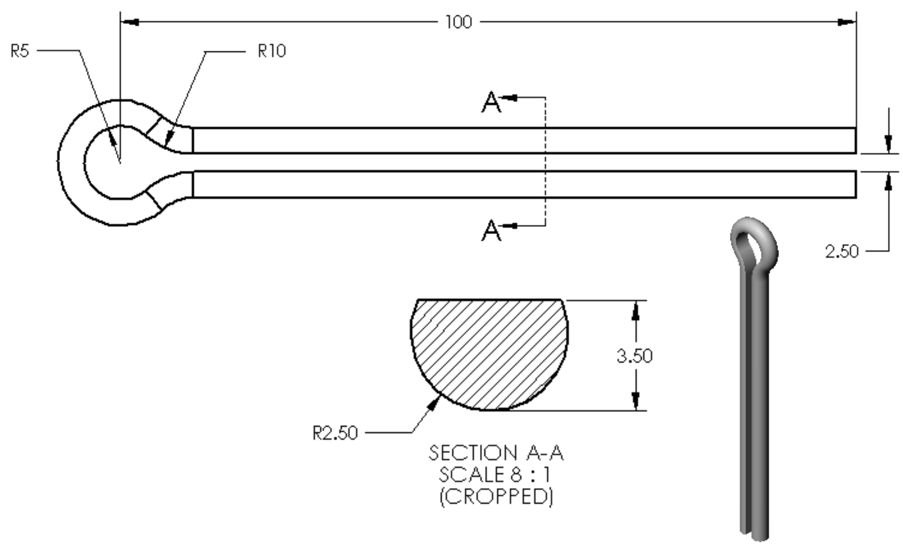
This lab uses the following skills:

- *Completing the Path and Profile Sketches* on page 202.
- *Introducing: Sweep* on page 203.

Units: **millimeters**

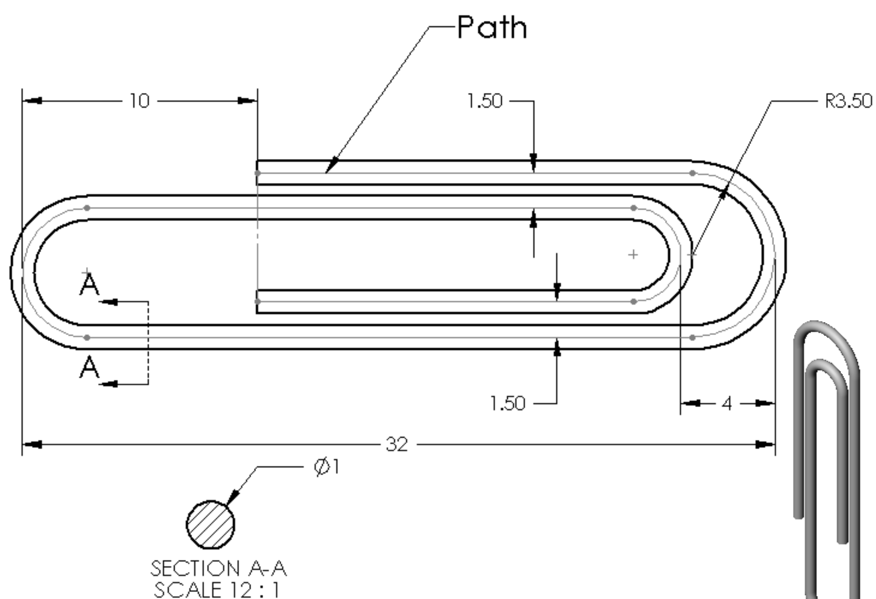
Cotter Pin

The `Cotter Pin` uses a path that describes the inner edge of the sweep.



Paper Clip

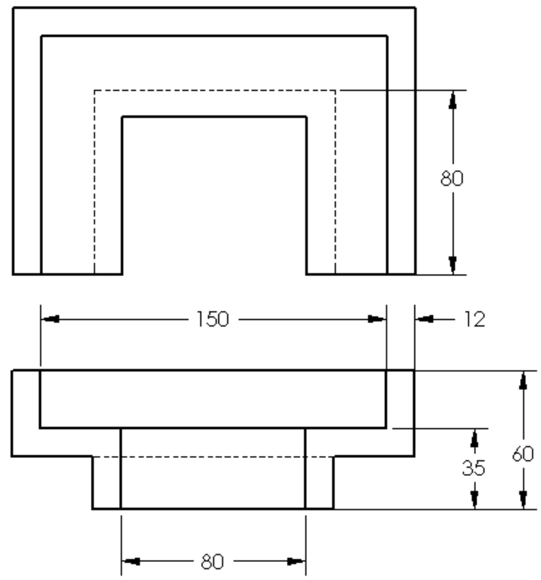
The `Paper Clip` is defined by a path that describes the centerline of the sweep.

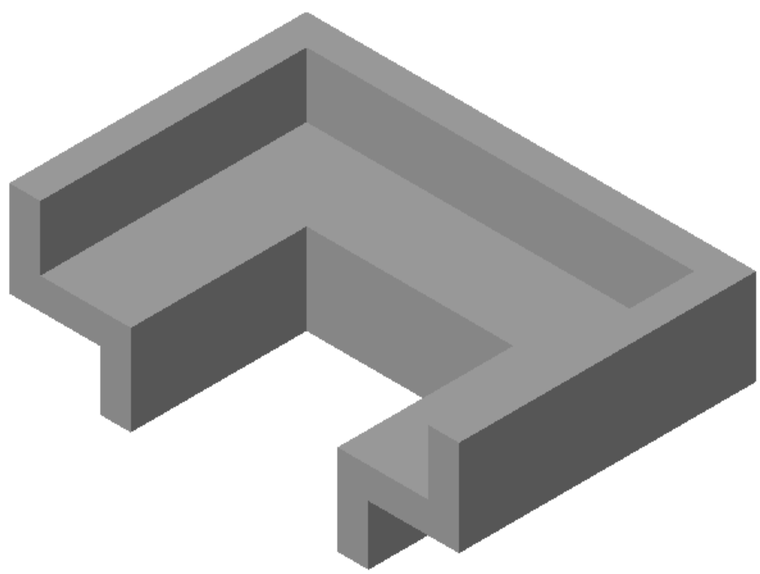


Thanks to Paul Gimbel, TriMech Solutions, LLC for submitting these examples.

Mitered Sweep

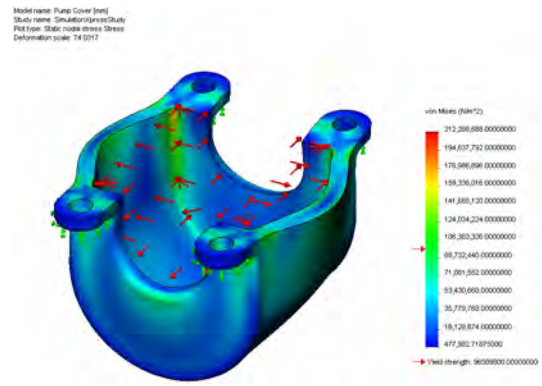
The `Mitered Sweep` is defined by a path that describes the outer edge of the sweep.





Exercise 30: Simulation-Xpress

Perform a first pass stress analysis on an existing part.



This lab uses the following SimulationXpress skills:

- *Phase 1: Fixtures* on page 219.
- *Phase 2: Loads* on page 220.
- *Phase 3: Material* on page 221.
- *Phase 4: Run* on page 221.
- *Phase 5: Results* on page 222.

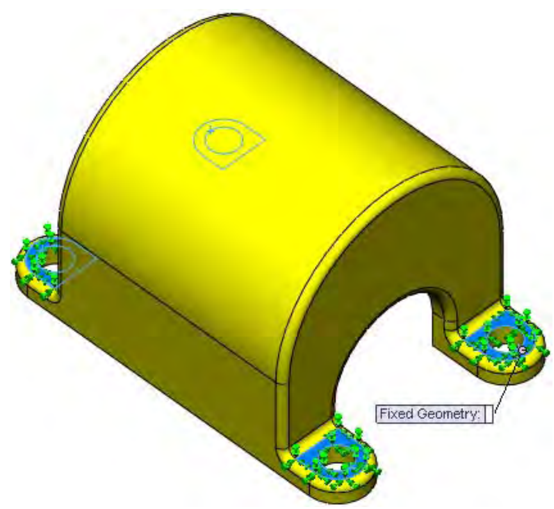
Units: **millimeters**

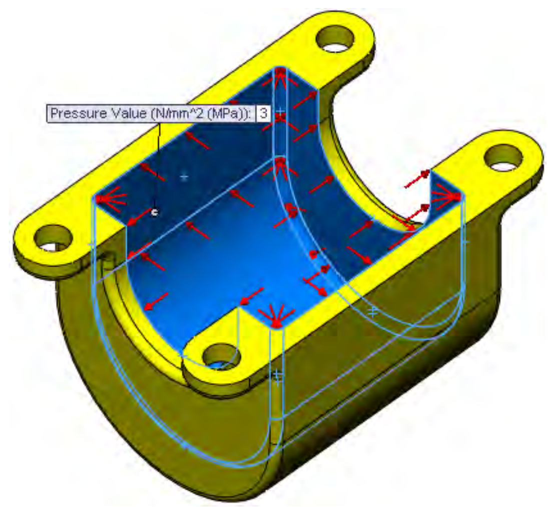
1 **Open Pump Cover.**
This part represents a cover that will be filled with oil under high pressure.

Start the SimulationXpress wizard.

2 **Set the units.**
Click **Options...** and set the units to **SI**.

3 **Define the fixture.**
Select the uppermost faces of the four tabs and the cylindrical faces of the four bolt holes.





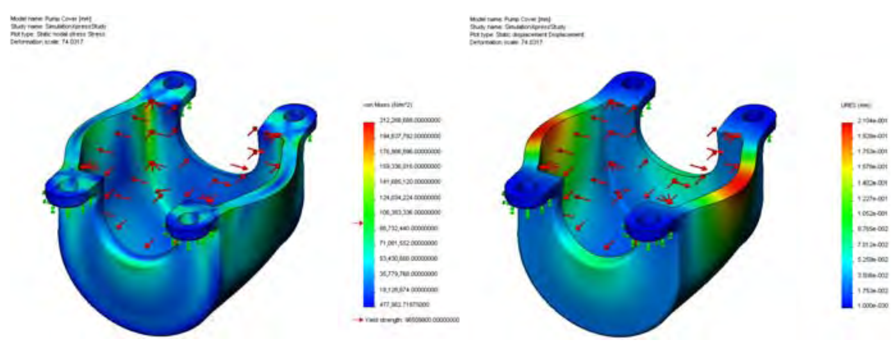
4 Define the load set.
Select **Pressure** for the type of load. Right-click one of the faces on the *inside* of the Pump Cover. Pick **Select Tangency** from the shortcut menu.

5 Set the pressure value and direction.
Set the pressure value to **250 psi**.

6 Specify the material.
Select **Aluminum Alloys** and select **2014 Alloy** from the list.

7 Run the simulation.

8 Results.
The factor of safety is less than 1 indicating that the part is over stressed. Also view the stress and deformation plots.



9 Change the material.
Right-click the Pump Cover (-2014 Alloy-) icon in the SimulationXpressStudy and choose **Apply/Edit Material**.

Change the material to **Other Alloys**, **Monel(R) 400**.

10 Update.
Click **Run Simulation** to rerun the analysis using the new material. The factor of safety should be greater than 1.

11 Save and close the part.

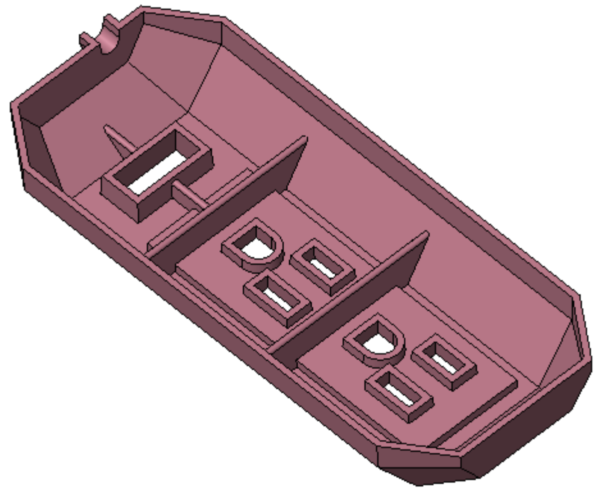
Lesson 7
Shelling and Ribs

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

- Apply draft to model faces.
- Perform shelling operations to hollow out a part.
- Create 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

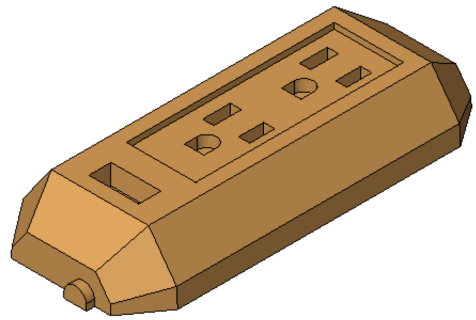
Some key stages in the modeling process of this part are given in the following list:

- **Draft with a plane**
  Draft can be defined with respect to a planar face or 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

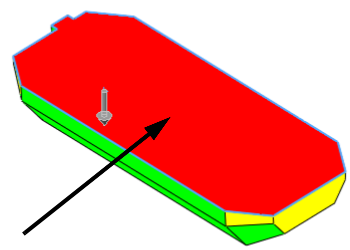
- Click **Draft Analysis** on the View toolbar.
- Or, click **View, Display, Draft Analysis...**.

Note

The dialog may vary slightly depending on your graphics card.

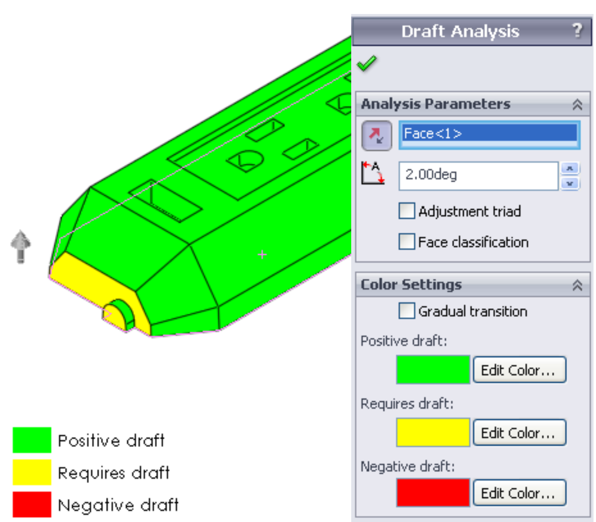
2 Click **Draft Analysis**.

3 Direction of pull.
Select the bottom planar face as the **Direction of Pull**. Click **Reverse Direction** so the pull direction arrow points as shown.



4 Results.
Set the **Angle** to **2°**. Colors are assigned to the faces according to the nature of their draft. Three (yellow) faces require draft.

Click **OK** to complete the command and the face colors and legend will remain visible.



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 base feature, there isn't any draft on it. There is 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

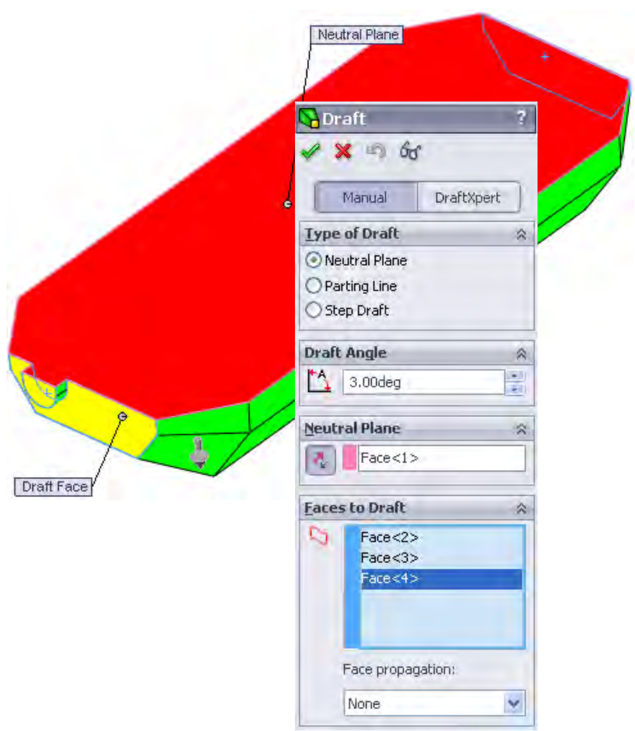
The process of adding draft requires the selection of one **Neutral plane** and one or more **Faces to draft**.

5 **Neutral plane draft.**
Click **Insert, Features, Draft...** and choose **Neutral Plane** as the **Type of Draft**.

Select the bottom planar face, same as for draft analysis, as the **Neutral Plane**.

Set the **Draft Angle** to **3** degrees.

Select the three yellow faces to draft and click **OK** to complete the command.

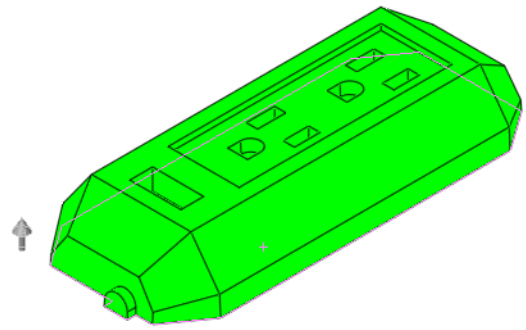


Note

Click **Reverse Direction** if necessary to point the arrow in the direction shown.

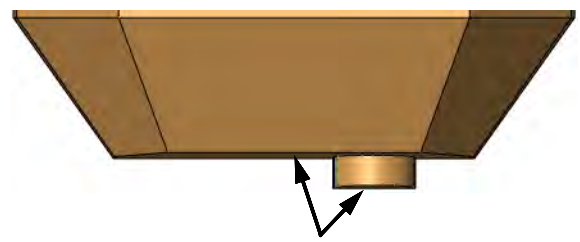
6 **Draft analysis recheck.**
Notice color change for the selected faces. They now have positive draft within the boundaries set in the draft analysis.

Click the **Draft Analysis** tool again to shut off the colors.



Tip

Zooming in on the Top view shows the drafted faces.



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.

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

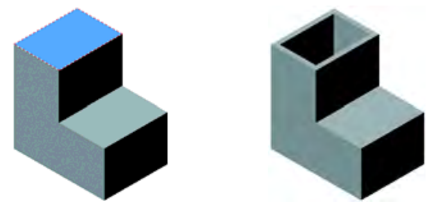
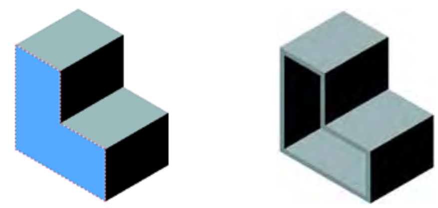
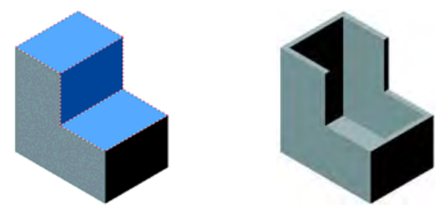
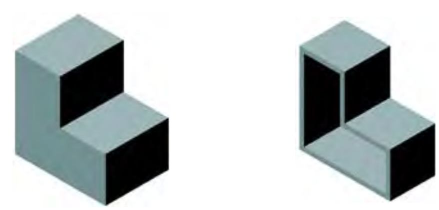
- From the **Insert** menu, choose **Features, Shell...**.
- Or, on the Features toolbar, click **Shell** .

Tip

Clear **Show preview** before selecting the faces, otherwise the preview will be updated with each selection, slowing down the operation.

Face Selection

Shelling can remove one or more faces from the model or create a fully enclosed void. Here are some examples:

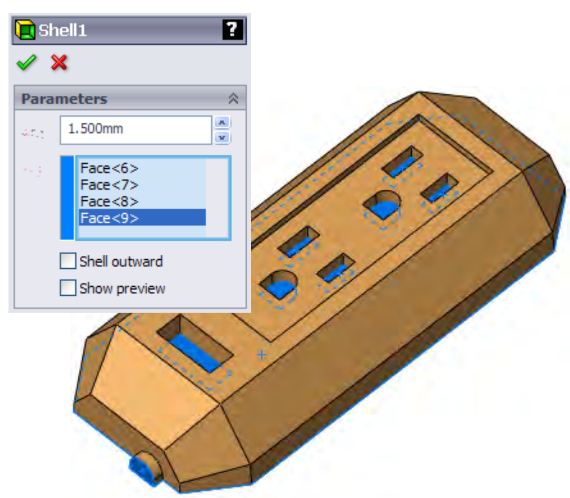
One face selected.	
One face selected.	
Multiple faces selected.	
No faces selected. Note: The results are shown sectioned, using the **Section View** command.	

7 Shell.

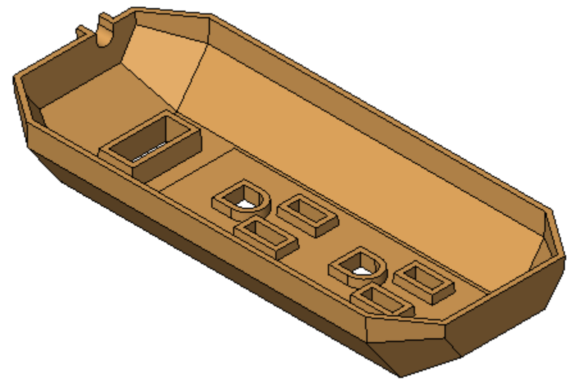
Click **Insert, Features, Shell...** and set the **Thickness** to **1.5mm**.

Clear **Show preview** and select the **9** faces (including the hidden bottom face) shown as the **Faces to Remove**.

Click **OK**.



8 **Shelled part.**
The selected thickness is applied to all faces except those removed.

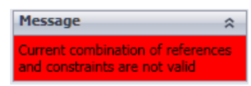


Planes

The **Plane Wizard** can be used to create a variety of planes using different geometry. Planes, faces, edges, vertices, surfaces and sketch geometry can all be used to apply constraints through **First**, **Second** and optionally **Third References**. The **Fully defined** state is listed when it is reached.

Tip

If the selections cannot be combined to form a valid plane, a message appears in the dialog.



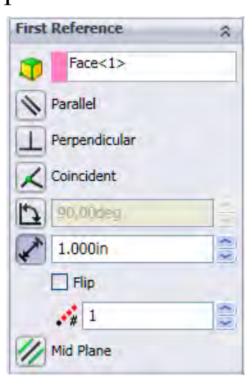
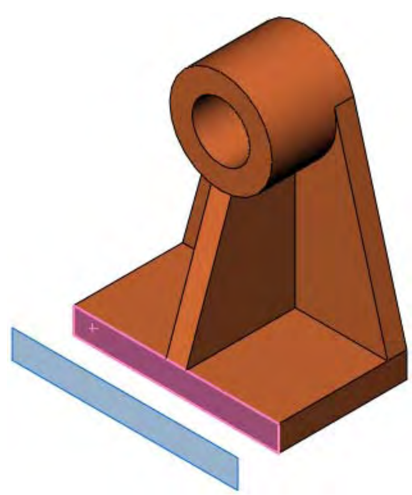
Where to Find It

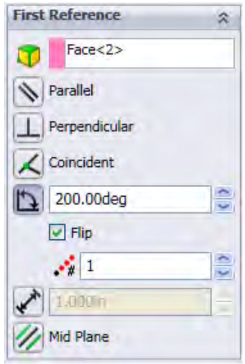
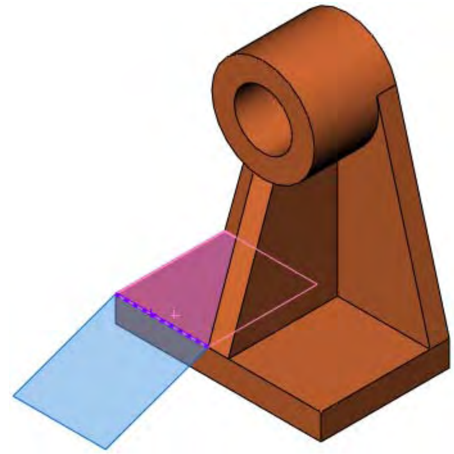
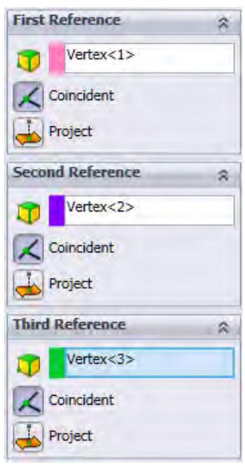
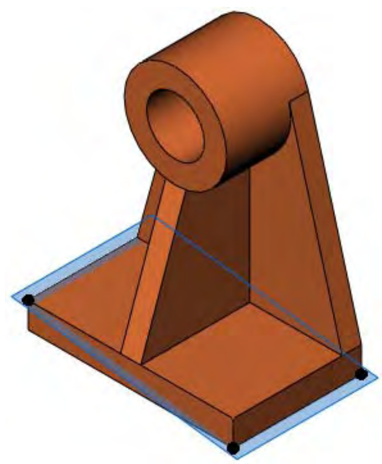
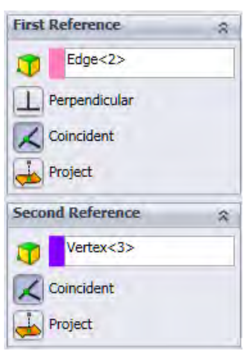
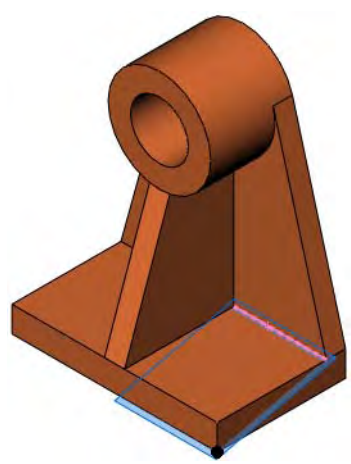
- Click **Plane** on the Reference Geometry toolbar.
- Or, click **Insert, Reference Geometry, Plane...**.

Shortcut

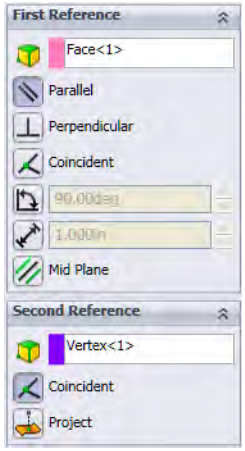
Press **Ctrl** and drag an existing plane to start the **Offset Distance** plane as shown below.

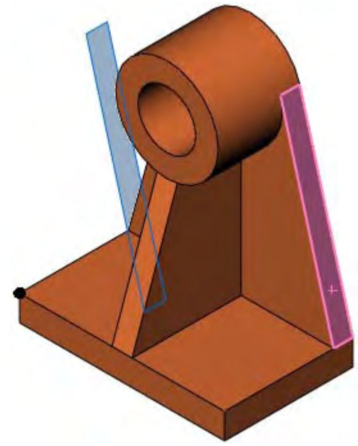
Here are some examples: Offset distance plane.

Offset Distance Select a planar face or plane and a distance.  	 Optionally create a series of parallel planes the same distance apart.

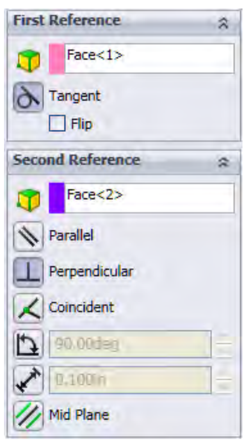
Angle Select a planar face or plane and an edge or axis.  	 Optionally create a series of angled planes the same distance apart.
Coincident Select three vertices.  	
Coincident Select a line and a single vertex.  	

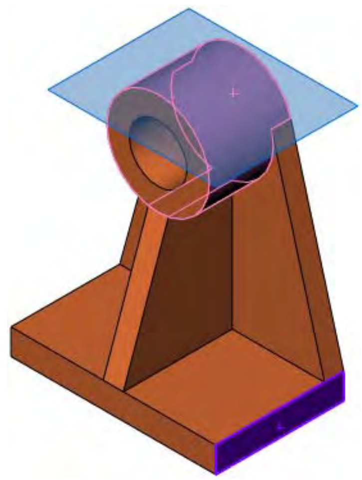
Parallel
Select a face and a vertex.





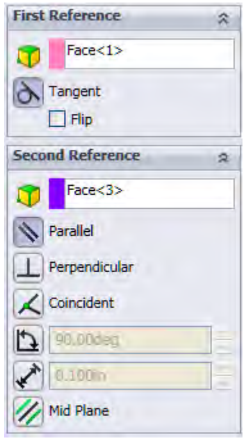
Tangent and Perpendicular
Select a cylindrical face and a planar face or plane with **Perpendicular**.

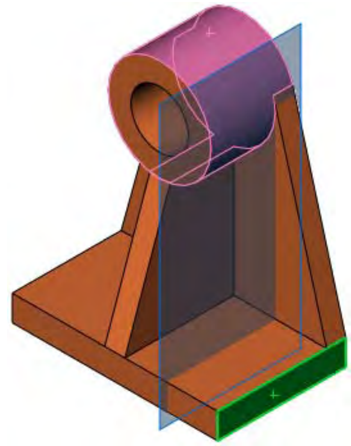




Tangent and Parallel

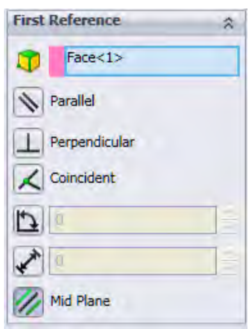
Select a cylindrical face and a planar face or plane with **Parallel**.

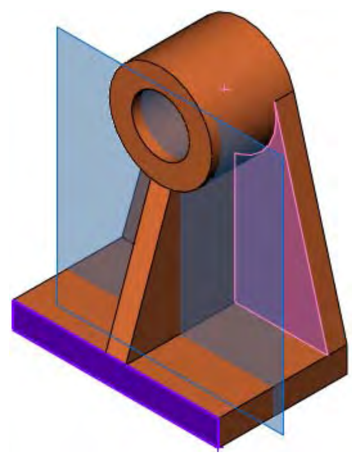




Mid Plane

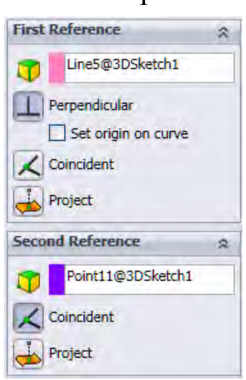
Select two planar faces with **Mid Plane**.

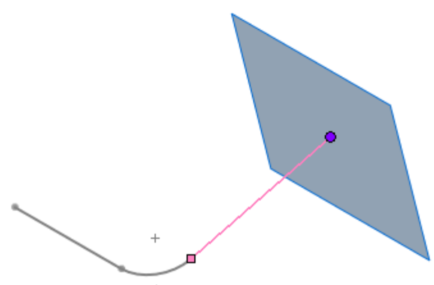




Perpendicular at a Point

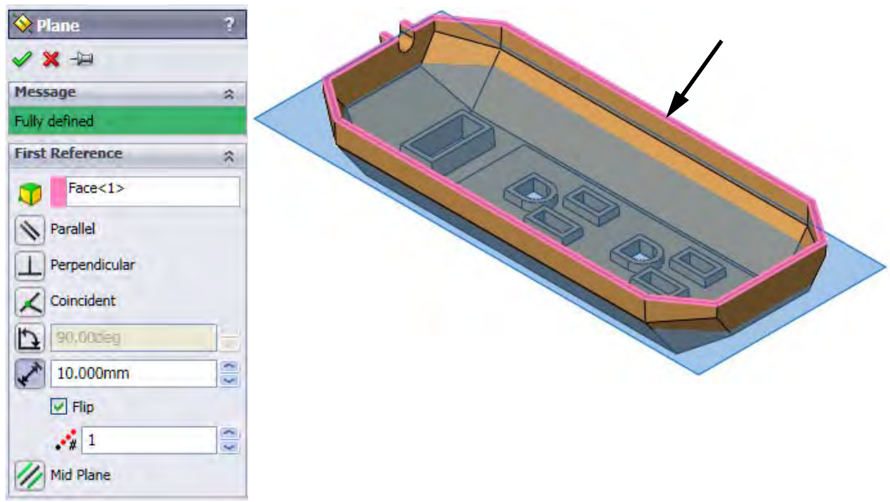
Select a sketched line and an endpoint.





As a shortcut to Perpendicular at a Point, select an edge/line and click **Insert, Sketch**. The plane is created and a sketch opened on it.

9 Offset distance plane.
Select the thickness face and use the **Plane** tool to create a new plane using an **Offset Distance** of **10mm** to the *inside* (intersecting the model).



Ribs

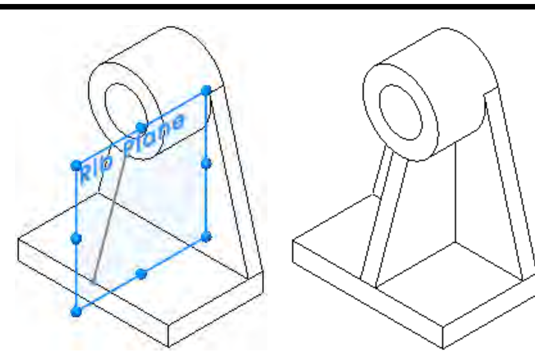
The **Rib** tool enables you to create ribs using minimal sketch geometry. The tool prompts you for the thickness, direction of the rib material, how you want to extend the sketch if necessary, and whether you want draft.

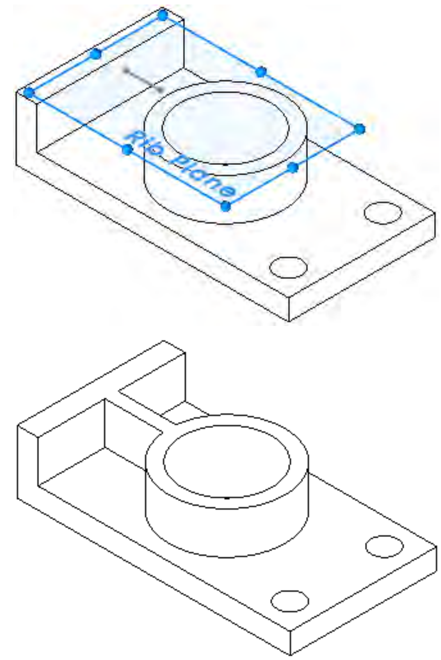
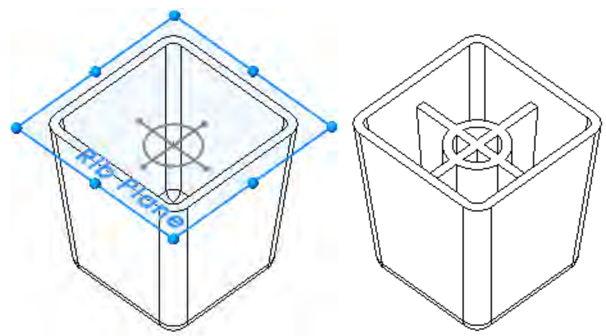
Tip

Unlike other sketches, the rib sketch does not have to cover the complete length of the rib feature. This is due to the fact that the rib feature automatically extends the sketch to the next feature it finds on both ends.

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 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.	

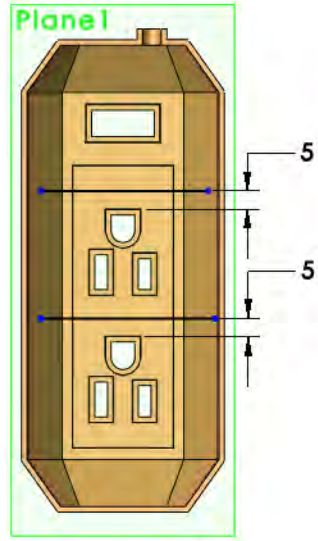
Introducing: Insert Rib

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

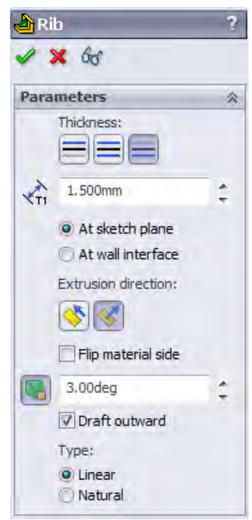
- From the **Insert** menu, choose **Features, Rib...**.
- Or the pick the **Rib** tool on the Features toolbar.

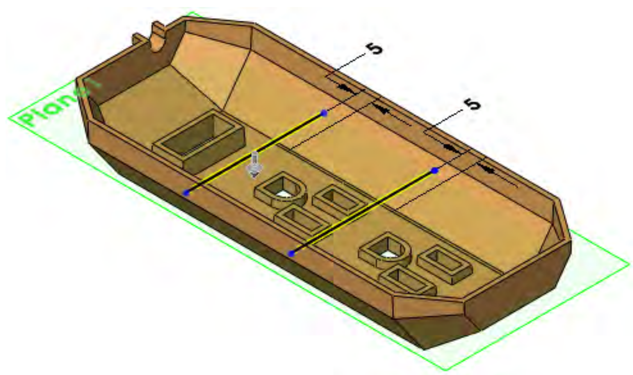
10 Sketch line.
Create a new sketch on the plane Plane1. Sketch two lines, under defined and dimensioned as shown. Note that the lines are **Horizontal**.



11 Rib tool.
Click the **Rib** tool and set the parameters shown:

- **Thickness: 1.5mm**
  Create rib on **Both Sides** of sketch **At sketch plane**
- **Extrusion direction: Normal to Sketch**
- **Draft : 3° outward**





Tip If the **Flip material side** arrow points away from the model, reverse the direction. Click **OK**.

12 Rib sketch.
Select the Right plane and create a new sketch. Change the display to **Hidden Lines Visible**.

Converting Edges

Convert Entities is used to create copies of model edges in the active sketch. The edges are projected onto the plane of the sketch, regardless whether they lie on that plane or not.

Introducing: Convert Entities

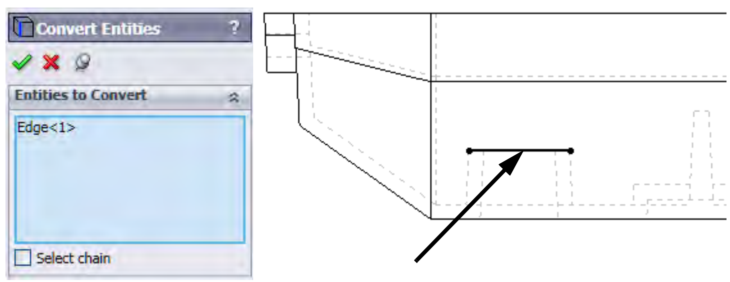
Convert Entities enables you to copy model edges into your active sketch.

Where to Find It

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

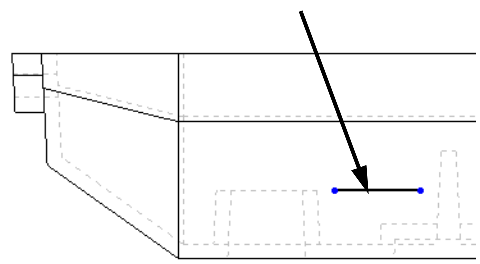
13 Convert.

Click **Convert Entities** and select the upper edge as indicated. Click **OK**.



14 Drag.

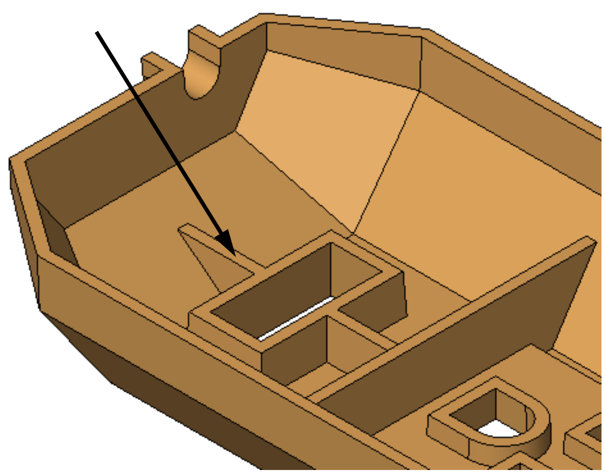
Drag the endpoints of the converted edge to move it to the right.



Use **Parallel to Sketch** with the options used in the previous rib feature to create the rib.

15 Completed ribs.

Another rib can be added using the same method as the previous one.

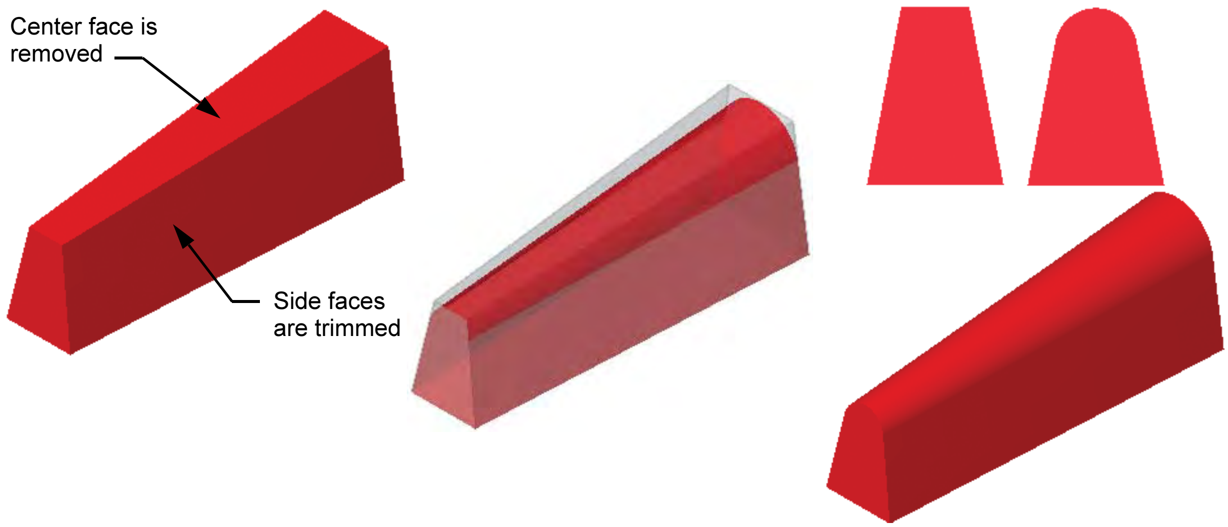


Full Round Fillets

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.



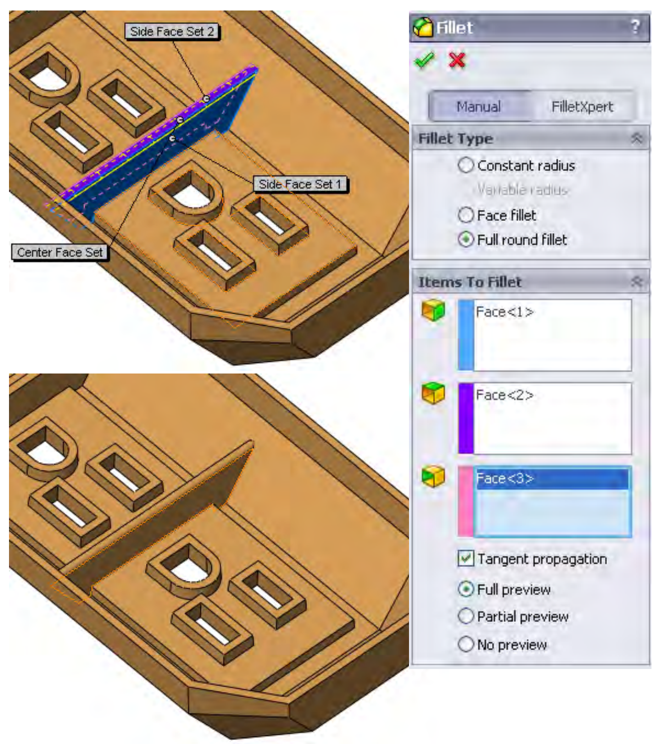
Where to Find It

- From the **Insert** menu, choose **Features, Fillet/Round...**.
- Or click **Fillet** on the Features toolbar.

---

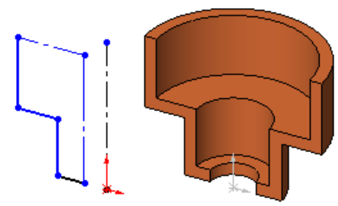
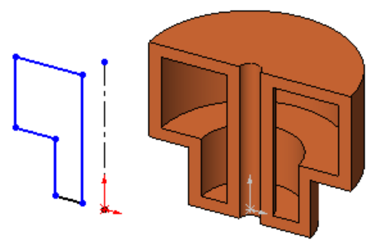
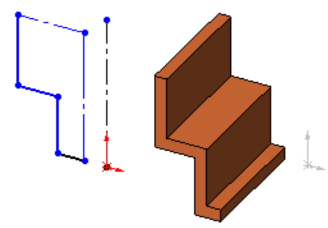
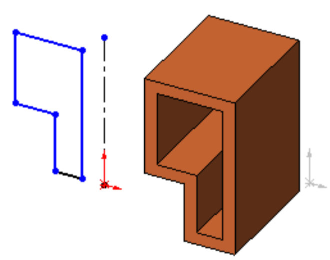
16 Full round.
Click **Fillet** icon and the **Full round fillet** option. Under **Items To Fillet**, select one face in each set as shown.

17 Save and close.



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.

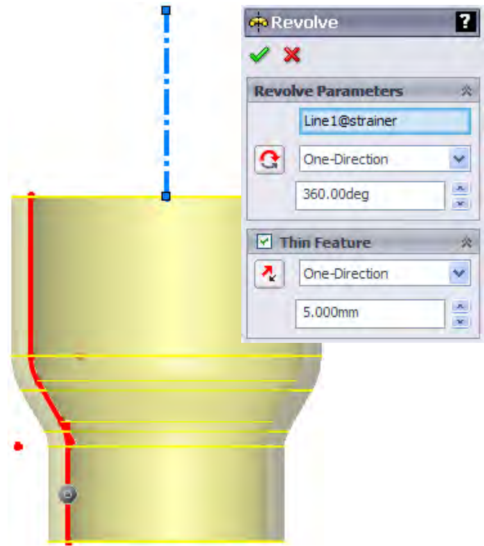
Revolve, open	
Revolve, closed	
Extrude, open	
Extrude, closed	

1 Open Thin_Features.

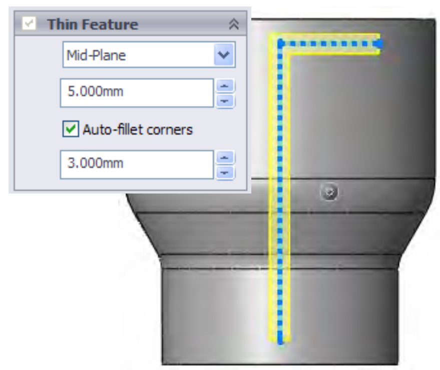
2 Thin revolve.
Select the strainer sketch and the **Revolve** tool. When the system asks whether the sketch should be automatically closed, click **No**.

Set the **Direction 1 thickness** to **5mm** and the direction to the outside.

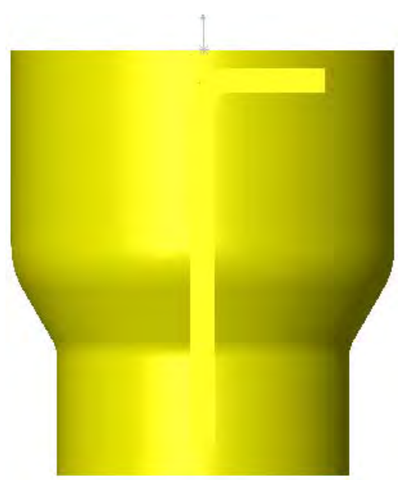
Click **OK**.



3 Thin extrude.
Select the bracket sketch and the **Extrude** tool. Set the **Thin Feature** to **Mid-Plane** and **5mm**. Click **Auto-fillet corners** and set the **Fillet Radius** to **3mm**.

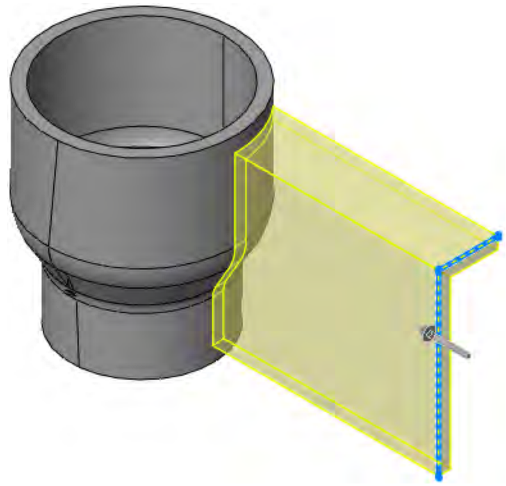


4 Preview.
Click **Detailed Preview** to view the auto fillets. Click the button again to dismiss the preview.



5 **Direction.**
Set the direction of the extrude towards the base feature and use **Up To Next**.

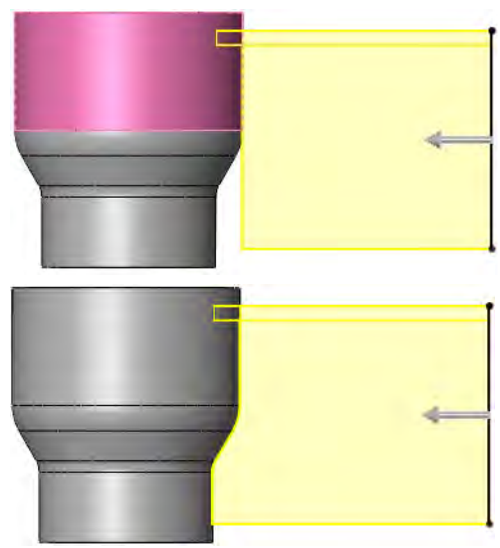
Click **OK**.



6 **Save and close the file.**

Note

This example offers another comparison between **Up To Surface** (top) and **Up To Next** (bottom).



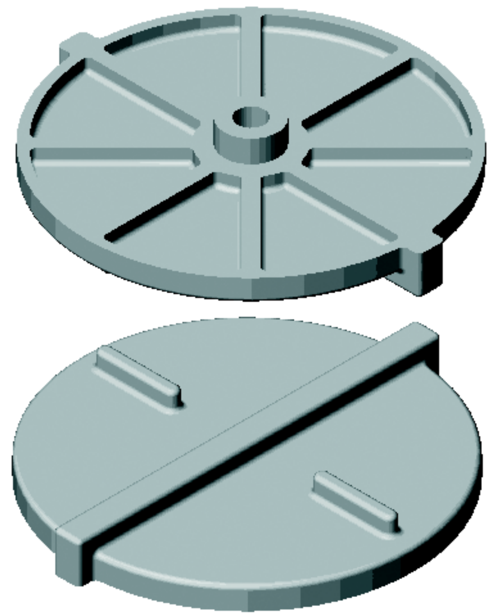
Exercise 31: Compression Plate

Create this part using the dimensions provided. Use relations wisely to maintain the design intent.

This lab uses the following skills:

- *Shelling* on page 243.
- *Ribs* on page 249.
- *Converting Edges* on page 252.

Units: **millimeters**



Design Intent

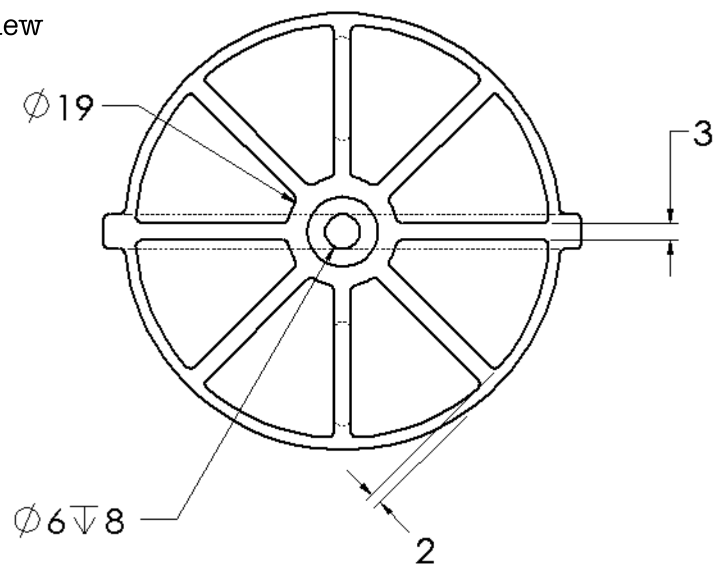
The design intent for this part is as follows:

1. Part is symmetrical.
2. Ribs are equally spaced.
3. All fillets and rounds are **1mm**.

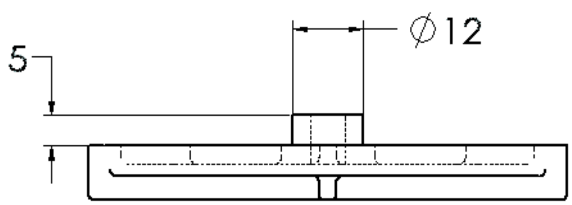
Dimensioned Views

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

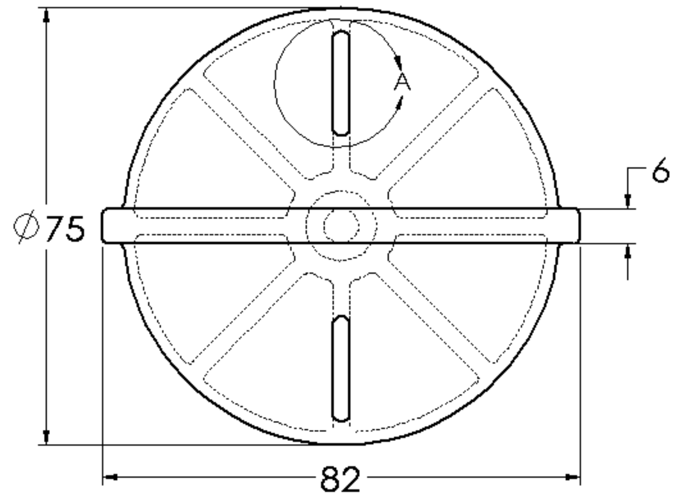
Top View



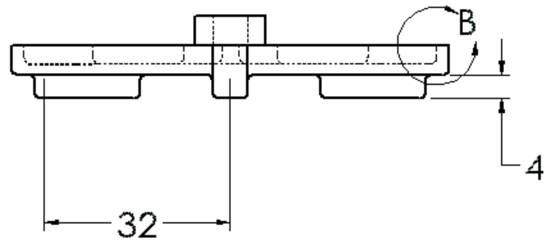
Front View



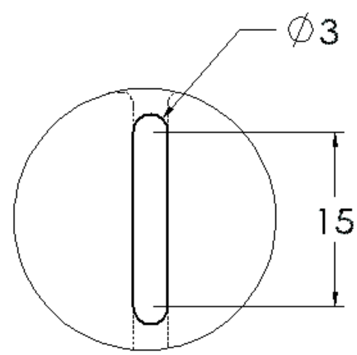
Bottom View



Right View

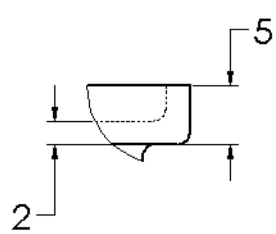


Detail A



DETAIL A
SCALE 4 : 1

Detail B



DETAIL B
SCALE 4 : 1

Exercise 32: Blow Dryer

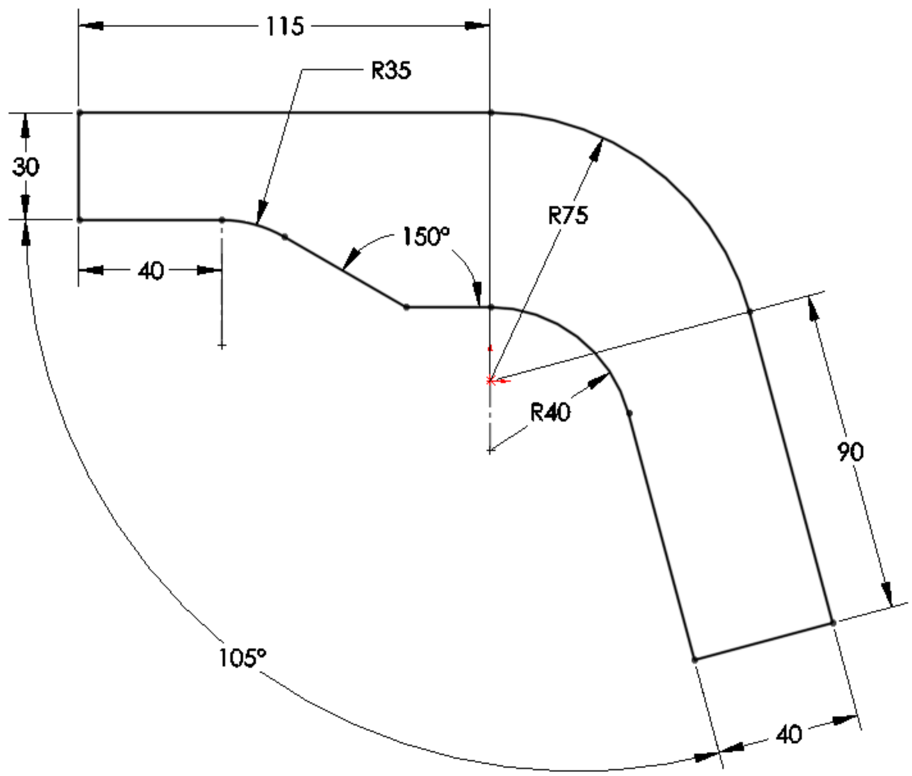
Create this part by following the steps as shown.

This lab uses the following skills:

- *Analyzing and Adding Draft* on page 241.
- *Shelling* on page 243.
- *Planes* on page 245.
- *Ribs* on page 249.
- *Full Round Fillets* on page 253.

Optional Sketching

If you would like to use the existing geometry, skip to **Procedure**. If you would like to create the sketch, open a new **mm** part and use the dimensions below. The sketch is on the Right Plane.



Procedure

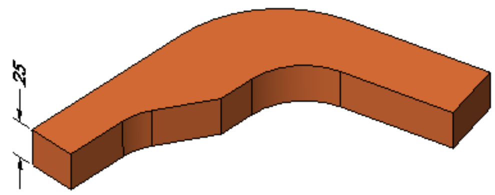
Open an existing part in the Exercises folder.

1 Open the part Blow Dryer.

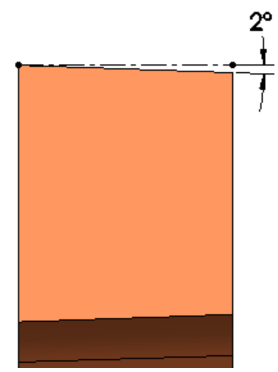
Extrude, Draft and Rounds

Starting with the sketch, the base feature is created, drafted and rounded.

2 Extrude.
Extrude the sketch **25mm** as shown.



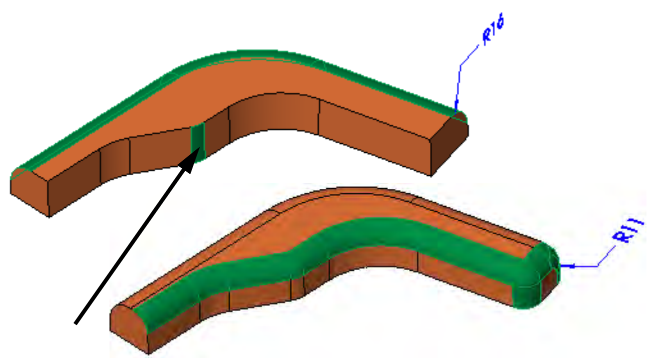
3 Draft.
Add draft of **2°** to all outer faces except the outlet face, using the back face of the model as the neutral plane. This is a partial Front view, looking *into* the outlet face.



Tip

There is no draft on outlet face.

4 Rounds.
Add rounds (**R16mm** and **R11mm**), using the size and sequence shown, to the solid body.



5 Check draft.
Using **Draft Analysis**, check the draft for **2°** against the Right Plane.

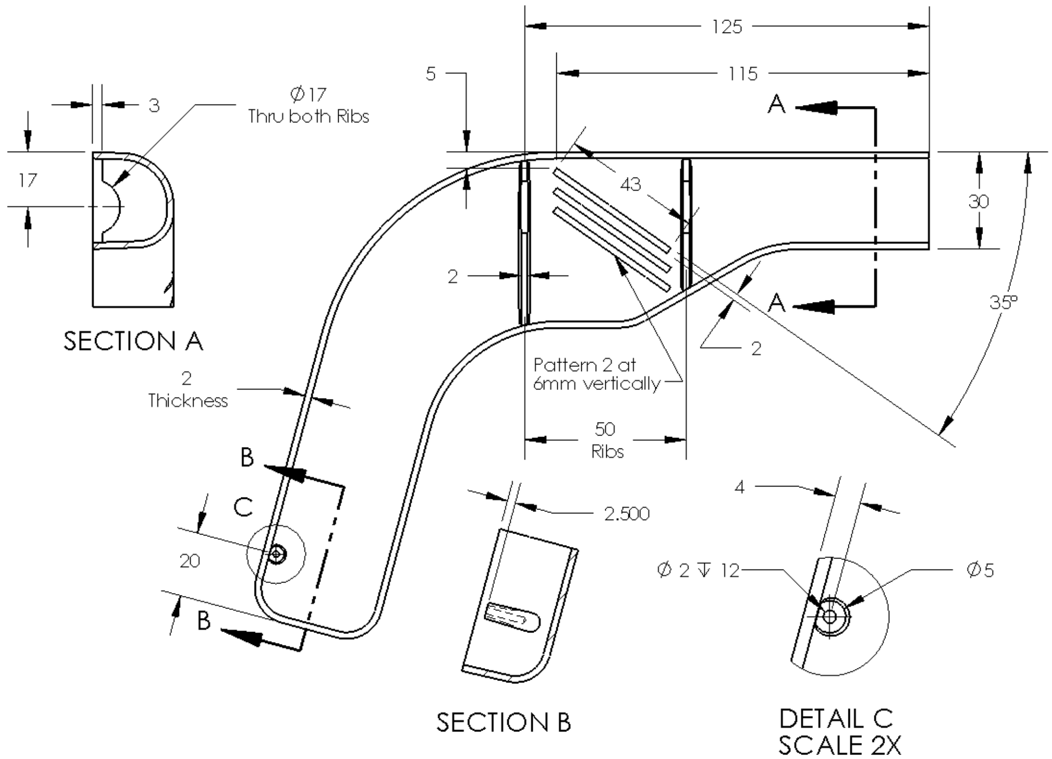
Other features

Use the following guidelines and drawing to add other features, including shells and ribs, to complete the model.

6 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.



7 Save and close the part.

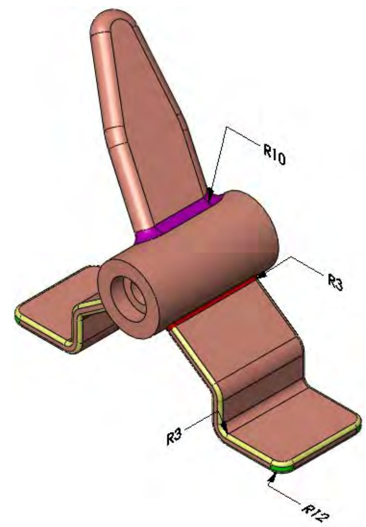
Exercise 33: Blade

Create this part using the information and dimensions provided.

This lab reinforces the following skills:

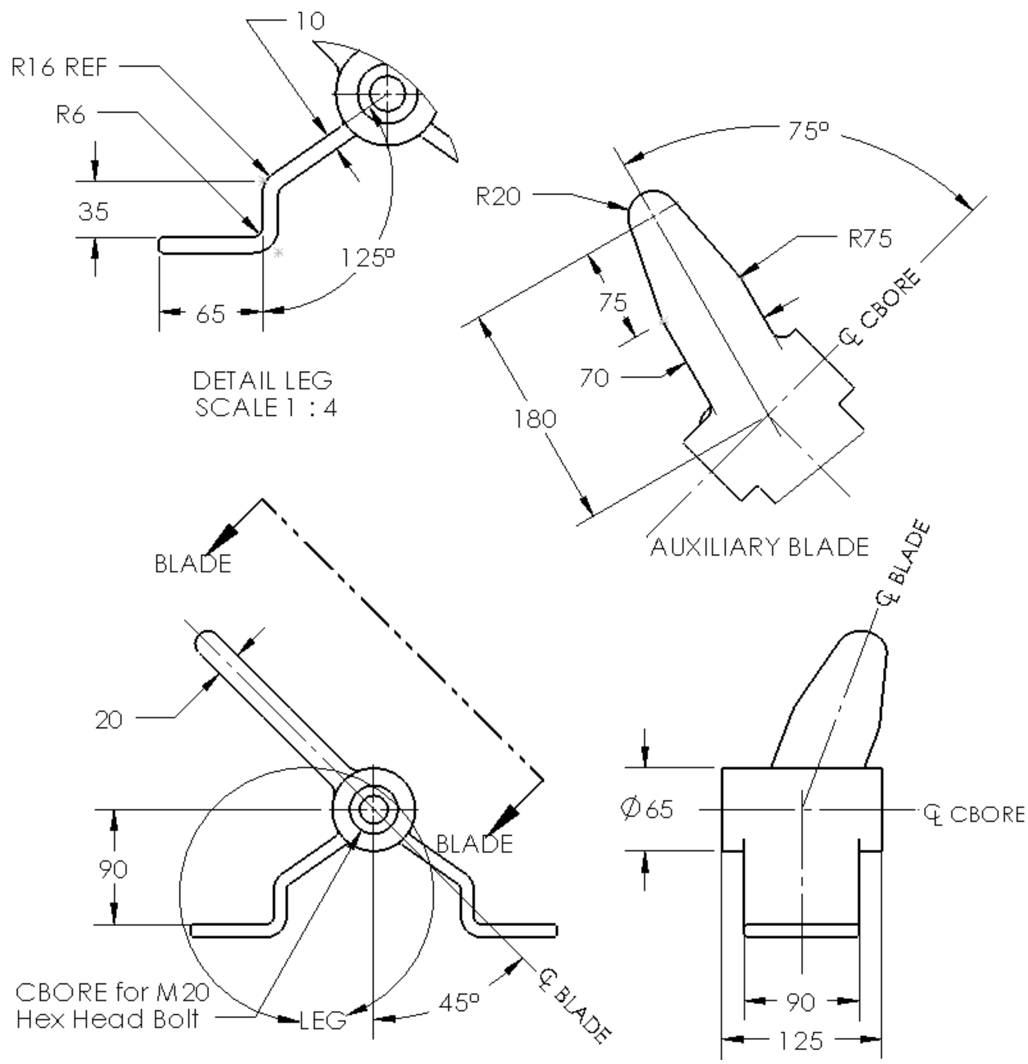
- *Planes* on page 245.
- *Full Round Fillets* on page 253.
- *Thin Features* on page 254.

Units: **millimeters**



Procedure

Create a new part. Refer to the image above for the sizes of fillets and rounds.



Lesson 8
Editing: Repairs

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

- Diagnose various problems in a part.
- Repair sketch geometry problems.
- Use the rollback bar.
- Repair dangling relations and dimensions.
- Use the FeatureXpert to repair filleting problems.
- Use the FilletXpert and DraftXpert to add fillets and draft.

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.

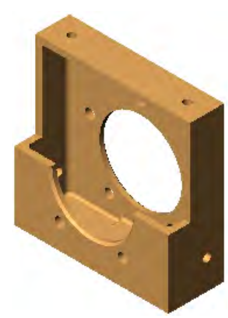
Stages in the Process

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 for Feature 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 PropertyManager that is used to create a feature is used to edit it.
- **Feature-, Fillet- and DraftXperts**
  Use the **FeatureXpert** to automatically repair errors in fillet and chamfer features. Use the **FilletXpert** and **DraftXpert** to add fillet and draft features.

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.

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.

Settings

Two settings in the **Tools, Options** dialog affect how errors are handled. **Show errors every rebuild** ensures that the error dialog appears after each rebuild. Use the **When rebuild error occurs** pull-down to control the action taken when a part is opened with errors. It can prompt for action, stop the rebuild at the error or continue.

Procedure

We will begin by making the proper settings.

---

1 **Error settings.**
Click **Tools, Options, System Options, General**. Click **Show errors every rebuild** and the **Prompt** pull-down from **When rebuild error occurs**. Click **OK**.

2 **Open the part named `Editing CS`.**
This part was built and saved with numerous errors.

3 **Feature failure.**
After opening, the system displays another message box, labelled **What's Wrong**. Each error is listed by feature name in the scrollable dialog.

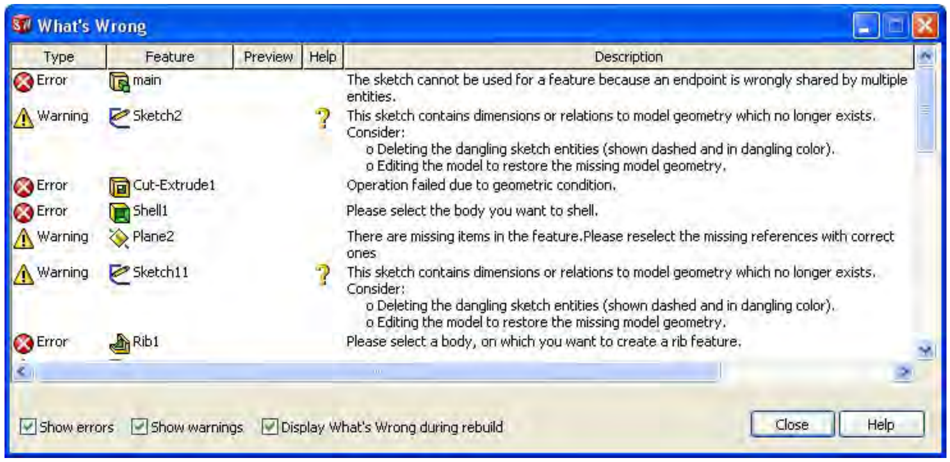
The model is not visible; errors have caused many features to fail.

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.

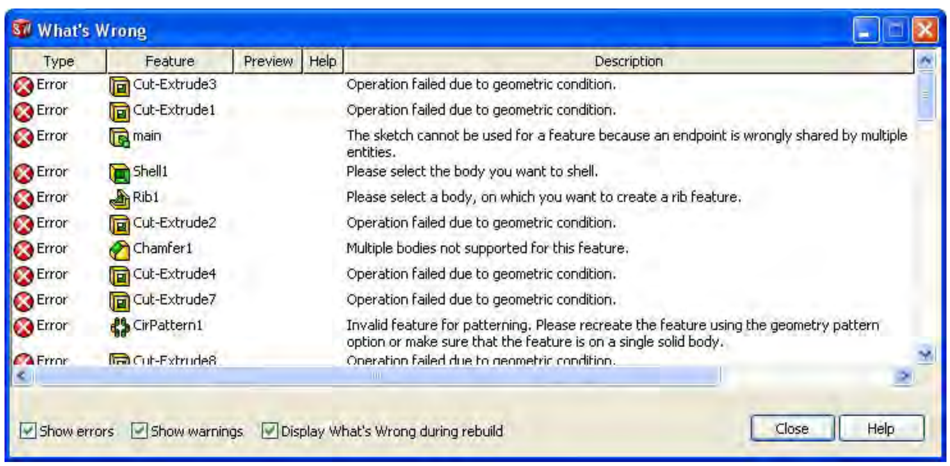
Tip

Click the question mark ? to open on-line help regarding this type of error.



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.



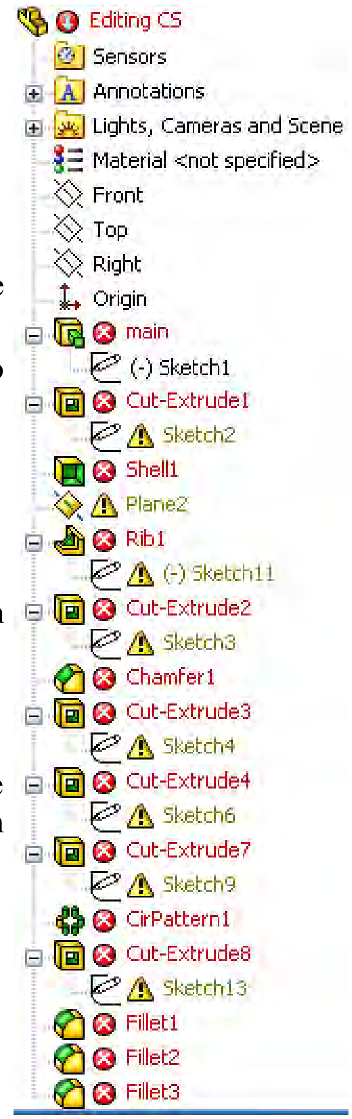
Note

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

4 FeatureManager design tree.

Close the **What's Wrong** dialog. The FeatureManager design tree lists many errors indicated with markers. The markers placed next to the features have particular meanings:



- **Top Level Error**

  The error marker next to the part name at the top of the tree 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 *yellow*.

- **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 *yellow*.

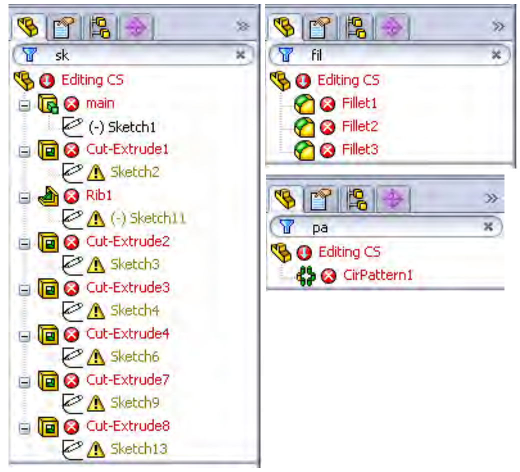
- **Normal Features**

  Normal features that do not have warnings or errors appear with *black* text.

Searching the Tree

The FeatureManager Search box  can be used to search by the starting letters of a name or some portion of the name. Try typing “sk”, “fil”, and “pa” as examples.

Click the “**x**” to clear the search.



Where to Begin

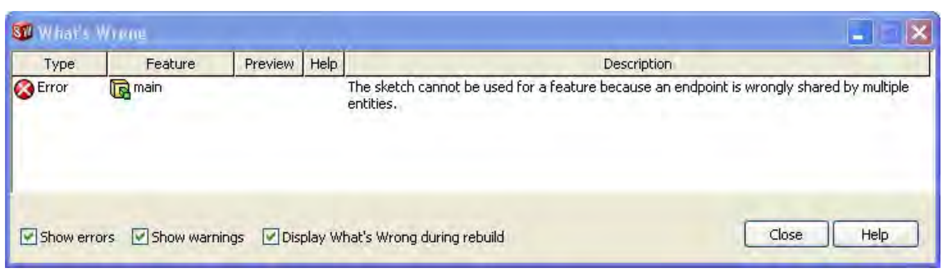
Features are rebuilt in sequence from the top of the tree. The best place to begin is at the first (base) feature with an error, in this case that is the feature main. An error in the base feature may cause a series of errors in the child features.

Many of the errors you will find are related to sketches. Don’t forget the *SketchXpert* on page 313 as a solution for over defined sketches.

---

5 What’s Wrong?

The **What’s Wrong** option is used to highlight an error message for a selected feature. Right-click the main feature and select **What’s Wrong?**. The message indicates that the sketch cannot be used for the feature because an endpoint is wrongly shared.



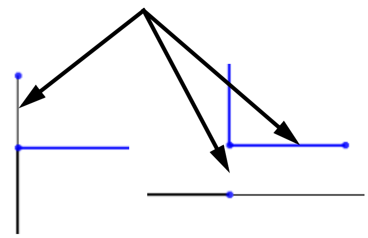
Tip

Floating the cursor over the error in the FeatureManager design tree will result in a balloon labelled with the feature name and the same description as the What’s Wrong description.

6 Edit the sketch.
The **What's Wrong** message has indicated the sketch (`Sketch1`) as the problem. Edit the sketch of the feature.

Sketch Issues

There are several reasons why sketches will not rebuild, but they can include geometry, relations or dimensions. Extra lines connected to existing endpoints and small unwanted pieces of geometry are common.

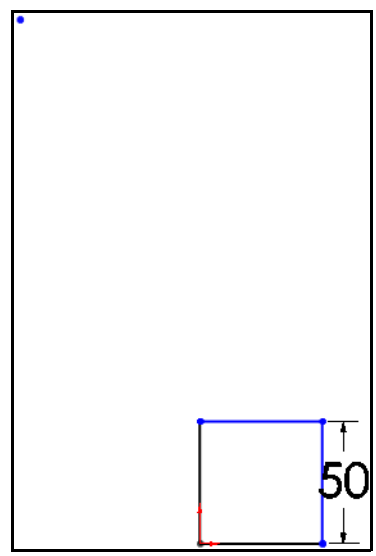


Note

A single gap in an otherwise continuous contour is acceptable.

7 Zoom to fit.
Geometry that is away from the intended profile geometry can cause sketch issues.

Click **Zoom to Fit** and all the geometry in the sketch will be shown. There is a very small piece of disconnected geometry.



Box Selection

Box selection enables 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.

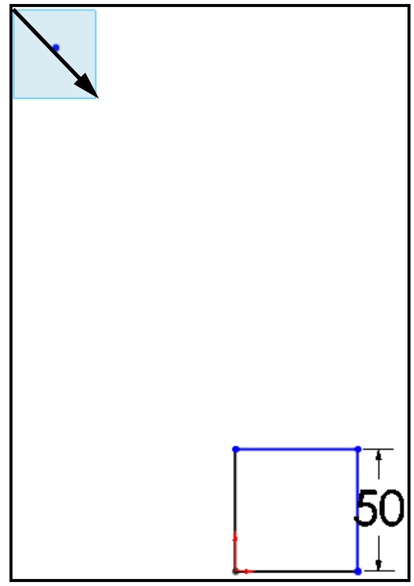
Right to left: The geometry within and crossing the window (short line and two long lines) is selected. This is also called **cross selection**.

Tip

Using **Shift** with the box selection maintains previous selections. Using **Control** with the box inverts the selections.

8 Select.
Use left to right box selection to select the extraneous line and delete it.

Zoom in on the remaining geometry.



Check Sketch for Feature

Check Sketch for Feature enables you to check the validity of a sketch for use in a feature. 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.

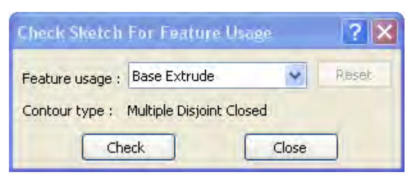
Where to Find It

- From the **Tools** menu, select **Sketch Tools, Check Sketch for Feature...**.

Note

If **Check Sketch** for Feature encounters sketch problems, it starts **Repair Sketch** automatically. See *Repair Sketch* on page 271.

9 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.

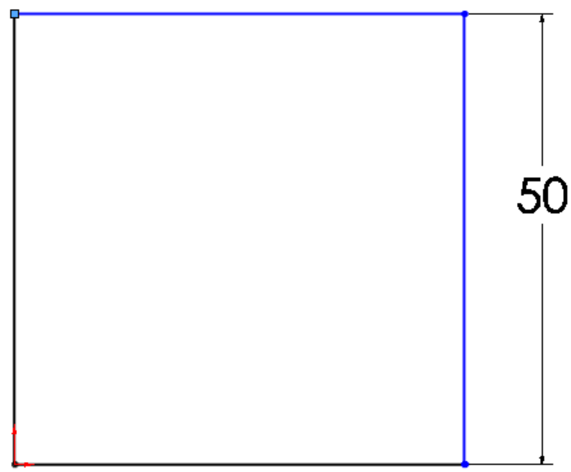
Click **Check**.

10 Message.

A message appears stating:

The sketch cannot be used for a feature because an endpoint is wrongly shared by multiple entities. To try to fix the sketch right now, click OK.

Click **OK**.



Repair Sketch

The **Repair Sketch** tool is used to find errors in the sketch and allow you to repair them. Repair sketch organizes the errors, describes them and zooms in on them using the magnifying glass.

Where to Find It

- Click **Tools**, **Sketch Tools**, **Repair Sketch**.

Magnifying Glass

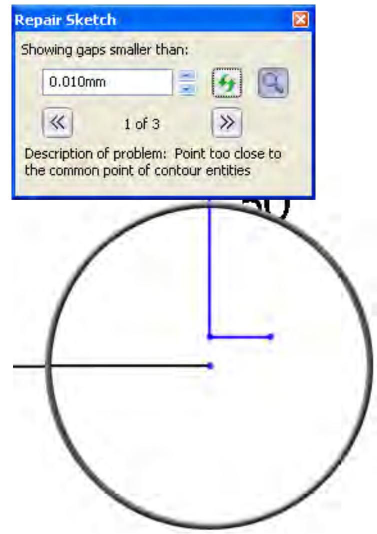
The **Magnifying Glass** tool is useful in finding and selecting small edges and faces in a model or an assembly. The magnifying glass typically is used while other tools are active. Some additional functions are:

- Use the middle mouse button/wheel to zoom within the glass.
- Use **Alt** + middle mouse button/wheel to section the model.
- Use **Ctrl** + middle mouse button/wheel to move the magnifying glass and pointer together.

Where to Find It

- Click the keyboard shortcut **g** to start the magnifying glass.

11 Repair Sketch.
Repair Sketch starts automatically. Set the **Gap** to **0.010mm** and click **Refresh** . Three errors appear with the magnifying glass zoomed in on the first one.



12 Next.

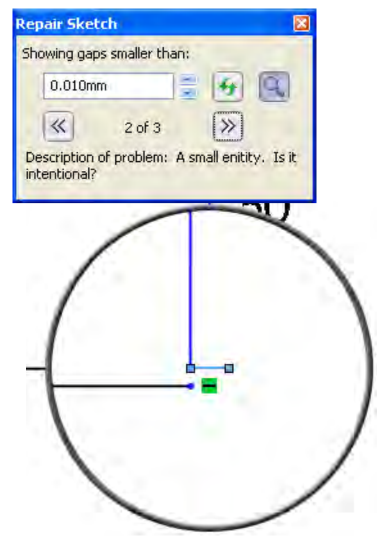
Click **Next** . Since two errors are in roughly the same area, the mirror remains in more or less the same place.

Rotate the wheel to zoom the magnifying glass in on the problem area.

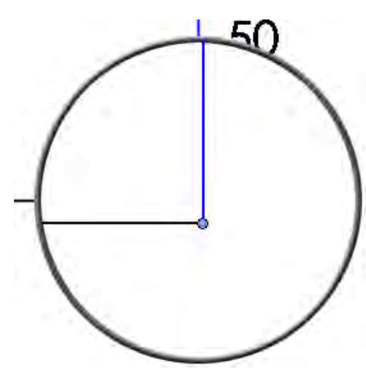
The error description is:

`A small entity. Is it intentional?.`

It isn't. Click the short line and delete it. Click **Refresh**.

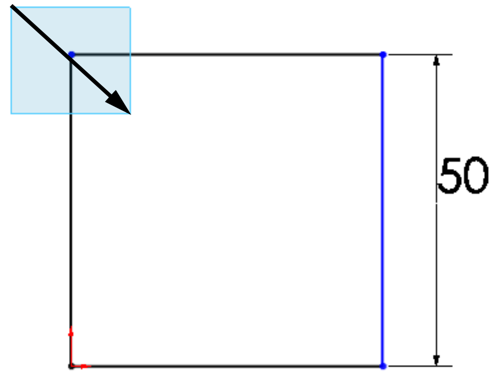


13 Two Points Gap.
The next error is listed as a `Two Points Gap`. As shown in the magnifying glass, there is a gap between endpoints. Select the endpoints and add a **Merge** relation.

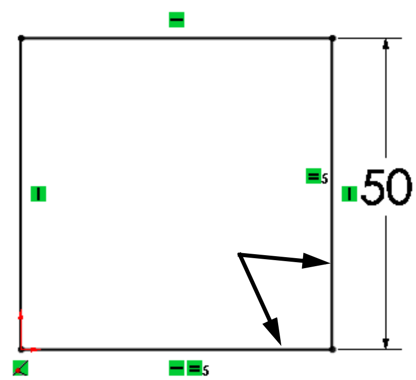


14 Last error.
Click **Refresh** and the last error appears. Close the dialog.

Use left to right box selection to select the extraneous line and delete it.



15 Equal.
Add an **Equal** relation between the edges as shown to complete the sketch.



Using Stop and Repair

You have the option to either rebuild the entire model to the end or stop at the next error and rebuild only to that feature. Choose one of two options at the message prompt.

Continue (Ignore Error) - Rebuilds the part and allows you to choose the next edit.

Stop and Repair - Stops at the next error and places the rollback bar after that feature. After each repair, SolidWorks will again stop at the next error.

16 Message.
Exit the sketch. A message appears to provide you with editing options:

`Feature Sketch2 has a warning, which may cause subsequent features to fail. Would you like to repair Sketch2 before SolidWorks rebuilds the subsequent features?`

Click **Stop and Repair**. The rollback bar is placed after the `Cut-Extrude1` feature.

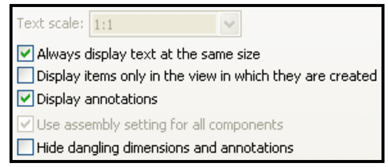
17 Next error.
The next error is listed in the What's Wrong dialog. So that you don't see this message dialog every time you make a correction, deselect the **Display What's Wrong during rebuild** option.

18 Next error.

The top error on the list is for `Sketch2` under the feature `Cut-Extrude1`. 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, Detailing**.



Reattach Collinear Relations

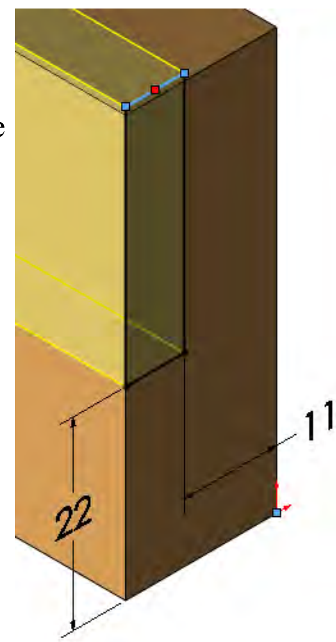
Dangling collinear relations can be quickly repaired by reattaching them to a similar *linear* edge of the model.

19 Edit Sketch.

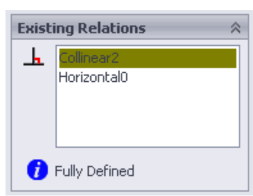
Right-click the `Cut-Extrude1` feature and choose **Edit Sketch**.

20 Dangling relations.

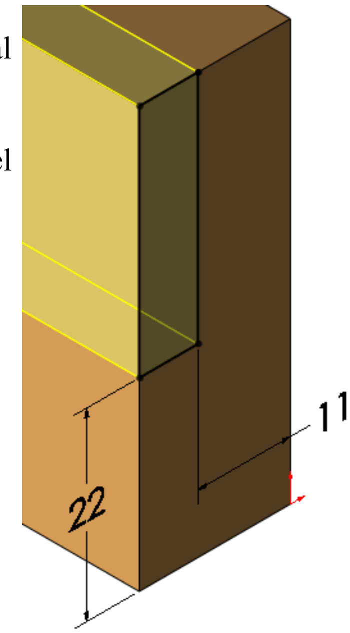
One line of the sketch is shown in the dangling color. Click on that line to select it and display its drag handles. The drag handle can be used in a drag and drop repair procedure.



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.



21 Reattach.
Drag the handle onto the topmost horizontal edge of the base feature. The system transfers the collinear relation from the missing entity (a 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 enables 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

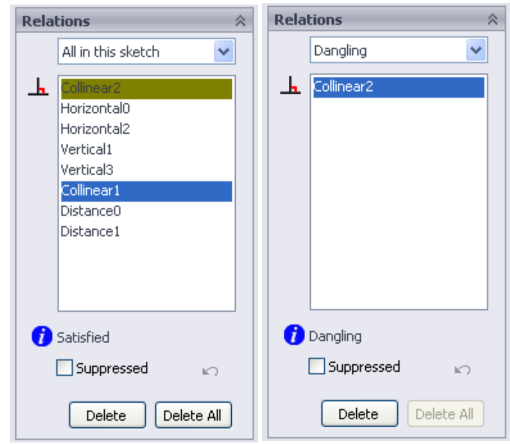
- Click **Tools, Relations, Display/Delete...**.
- Right-click in the sketch, and select **Display/Delete Relations**.
- Click **Display/Delete Relations** on the Sketch toolbar.

22 Undo.
Click **Undo** to remove the last event, the repair of the dangling relation.

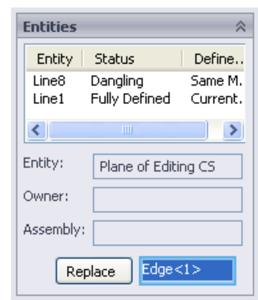
23 Display/Delete Relations.
Click **Tools, Relations, Display/Delete Relations.**

In the **Filter** list select **Dangling**. This displays only the relations that are dangling.

Select the **Collinear** relation.



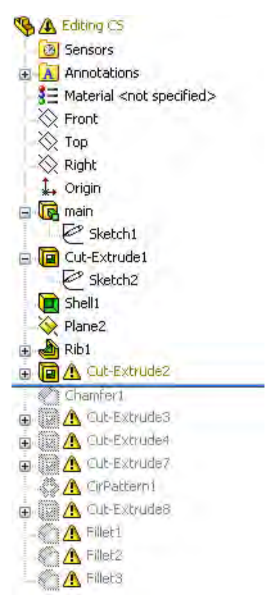
24 Entities section.
Look at the lower section of the PropertyManager. There is a list of the entities used by this relation. One entity has a **Fully Defined** status, the other is **Dangling**.



25 Replacement.
Select the entity marked **Dangling** and select the same top horizontal edge of the base feature as in step **21** on page 275.

Click **Replace** and then click **OK**, and close the sketch.

26 Roll forward.
Drag the rollback bar to a position between `Cut-Extrude2` and `Chamfer1`.



27 Message.
Right-click `Sketch3` of the `Cut-Extrude2` feature and choose **What's Wrong**.
The message states:
`The plane used by this sketch is missing and cannot be accessed. You can use command 'Edit Sketch Plane' to add a reference plane for this sketch.`

Repairing Sketch Plane Issues

Another common error occurs when a plane or feature with a planar face is removed that was used as a sketch plane for another feature. In this case, the sketch has an error and requires that a new sketch plane be assigned.

Introducing: Edit Sketch Plane

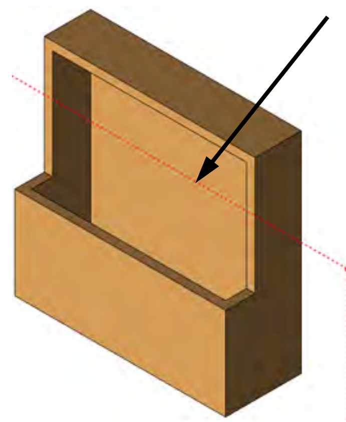
Edit Sketch Plane enables you to change the plane or face that a specific sketch is created on. The new sketch plane does not have to be parallel to the original.

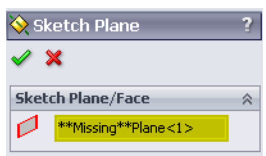
Where to Find It

- From the **Edit** menu, select **Sketch Plane...**.
- Or, right-click the sketch **Edit Sketch Plane** .

28 Edit sketch plane.

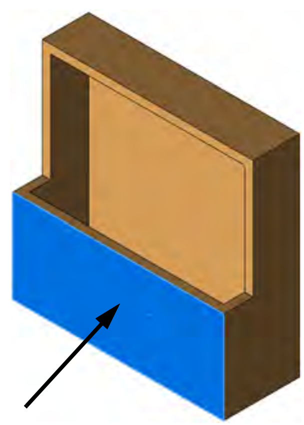
Right-click `Sketch3` and choose **Edit Sketch Plane** . The PropertyManager lists `**Missing**Plane<1>` as the sketch plane and a dashed outline of the missing reference is shown.





29 Select a replacement.

Select the front planar face of the part as a replacement. Click **OK**. The feature is repaired.



30 Roll forward.

Roll forward to just after `Cut-Extrude4`, which carries the next error marker. Use **What's Wrong** on this feature.

Reattach Dimensions

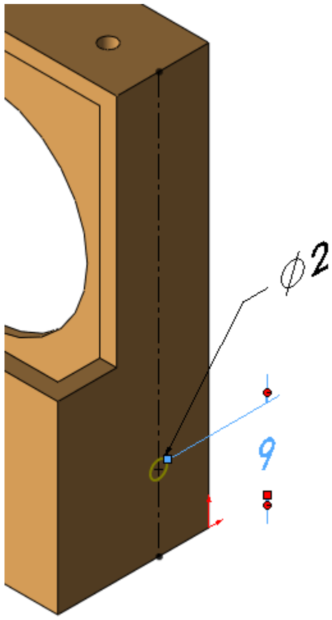
Dangling dimensions can be quickly repaired by reattaching them to edges or vertices of the model. The dimension value will reflect the new distance.

Tip

Dimensions can be reattached in this manner whether they are dangling or not.

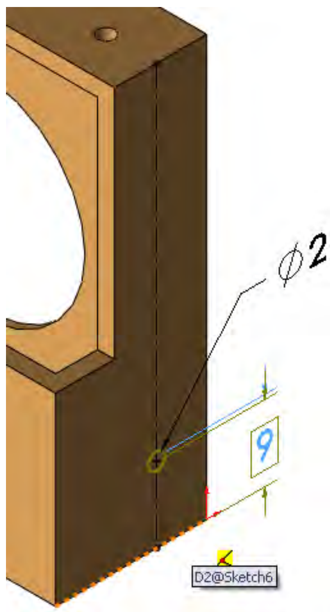
31 Edit the sketch.
Edit the sketch of the `Cut-Extrude4` feature. Note that the **9mm** dimension is the dangling color for dimensions and relations. The dimension is trying to attach to geometry that no longer exists, and therefore it is considered dangling.

Click on the **9mm** dimension to see the drag handles. The end marked with the red square handle is the dangling end, similar to how dangling relations are marked.



32 Drag and drop.
Drag the 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.

33 Exit the sketch to rebuild the model.

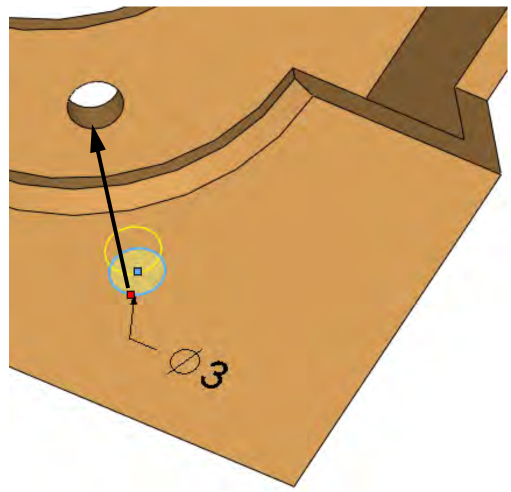


Reattach Concentric and Coradial Relations

Dangling concentric and coradial relations can be quickly repaired by reattaching them to a similar *circular* edge of the model.

34 Roll Forward.
Right click on `Cut-Extrude8` and select **Roll Forward**.

35 Edit sketch.
Edit the sketch of the `Cut-Extrude8` feature. Click the rightmost dangling circle (Concentric) to see the drag handle. Drag and drop the handle on the small circular edge behind. Repeat the procedure for the leftmost dangling circle (Coradial). Exit the sketch.

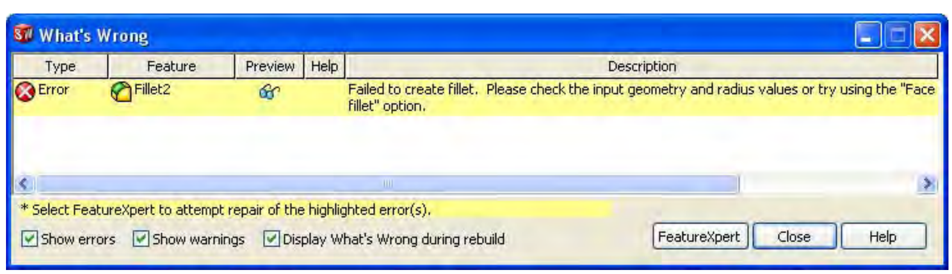


36 Roll to end.
Right-click in the FeatureManager and select **Roll To End** to position the rollback bar at the end of the FeatureManager design tree.

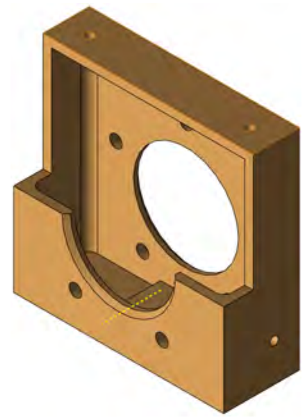
Highlighting Problem Areas

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.

37 Highlight message.
Click on the preview symbol to visually display the area in which the error occurs.



38 Graphic error display.
The area where the error occurs is highlighted with an edge. The fillet fails in the area indicated by the dotted line.



FeatureXpert

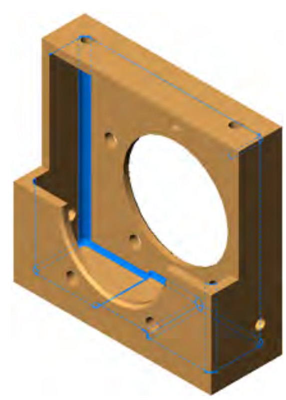
The **FeatureXpert** option is available for certain conditions where fillets or draft features fail. In this example, the `Fillet2` feature is failing. FeatureXpert will make use of all adjacent fillets to create a solution. The automated solution may involve splitting selections into different fillet features and reordering the sequence. The FeatureXpert is based on SolidWorks SWIFT technology.

Note

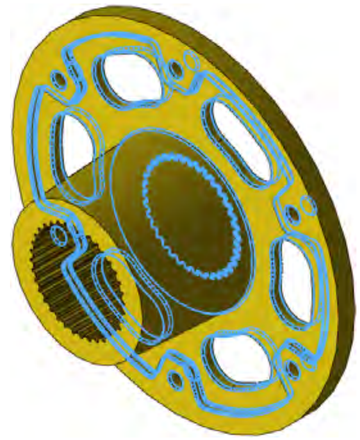
The option **Enable FeatureXpert** must be clicked from **Tools**, **Options**, **System Options**, **General** for this to be available.

39 FeatureXpert.
Click the **FeatureXpert** button on the What's Wrong dialog box. Note that the original fillet has been split into three fillets.

40 Model rebuilt.
The model is now rebuilt without any error or warnings. **Save** and close the model.



FilletXpert



In addition to solving fillet problems, the **FilletXpert** tool is used to create multiple fillets quickly and efficiently, leaving concerns of sequence to the system. It automatically leverages the **FeatureXpert** and **Reorder** to fix potential problems as you create the fillet features.

Options within the dialog allow you to **Add**, **Change** and **Remove** fillets.

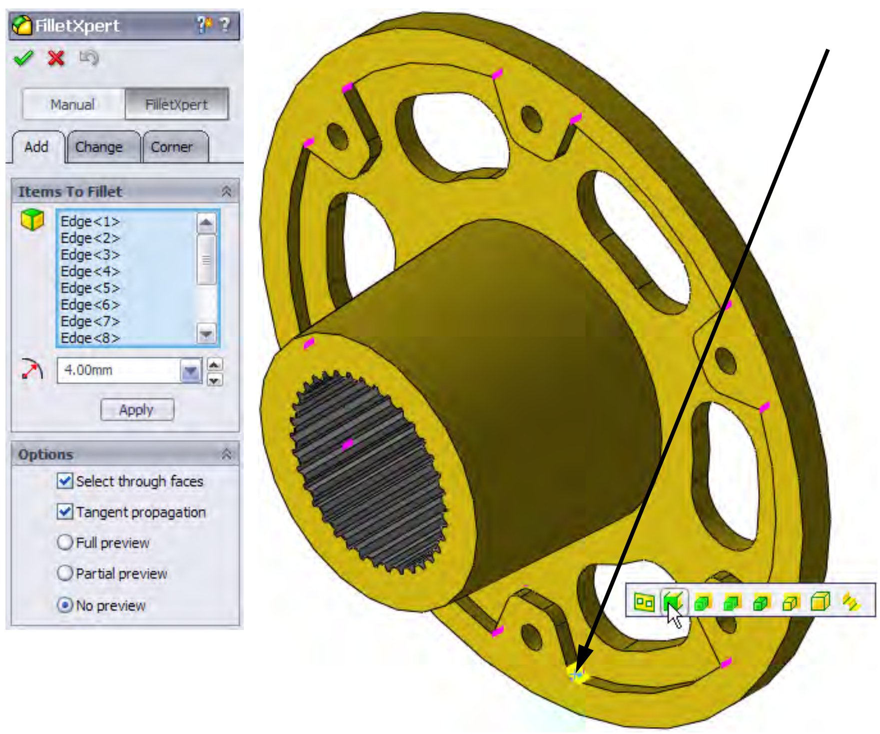
The **FilletXpert** tool is accessed from the standard **Fillet** tool.

---

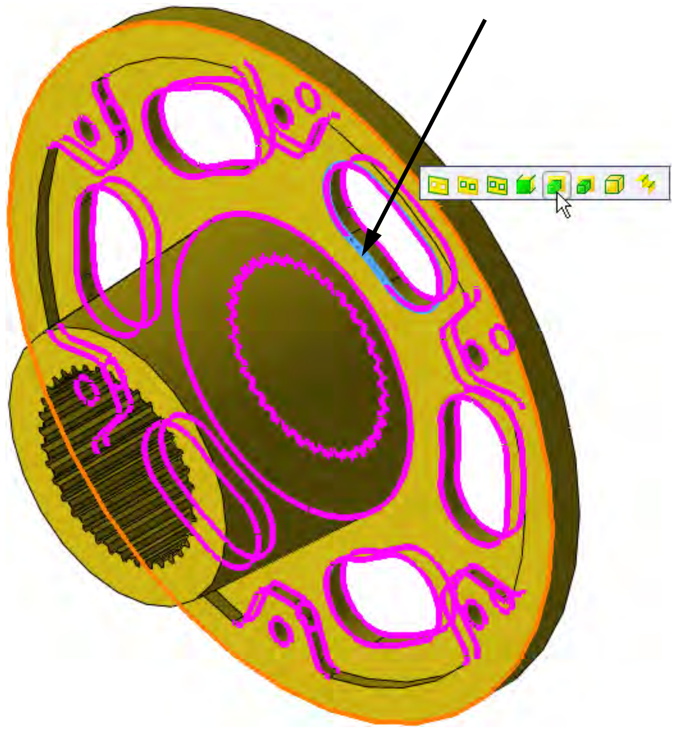
1 Open the part FilletXpert.

2 FilletXpert.

Click **Fillet** and **FilletXpert**. Set the radius to **4mm**. Select the edge as shown and the **Connected to start loop** option. Click **Apply**.



3 Edge selection.
Set the radius to **1mm.** Select the edge shown and the **Between left feature and part** option. Click **Apply**.



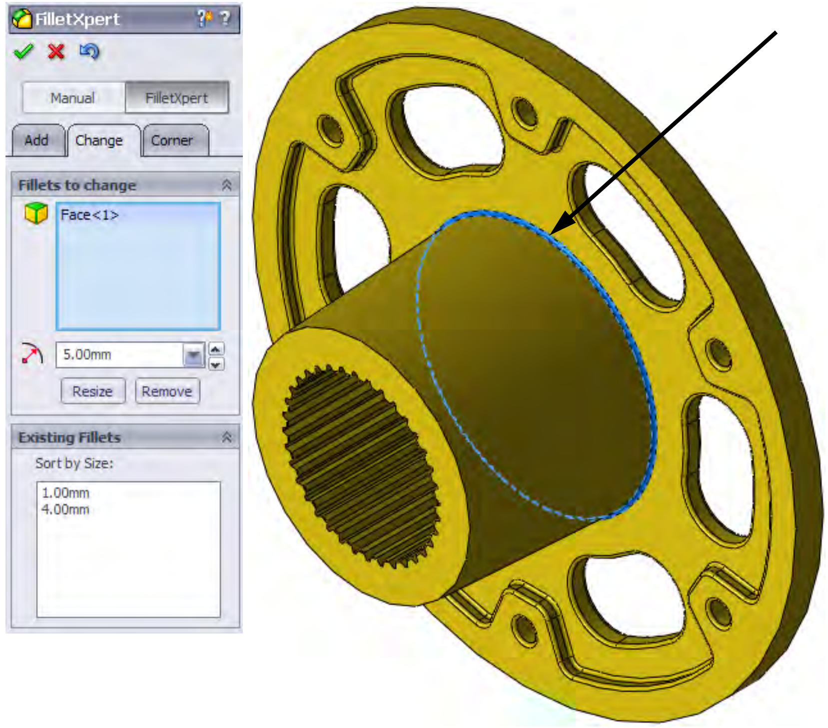
Changing Fillets

Using the **Change** tab on the **FilletXpert** dialog allows you to **Resize** or **Remove** fillets within the context of the PropertyManager. This allows you to make bulk selections and edit individual fillets.

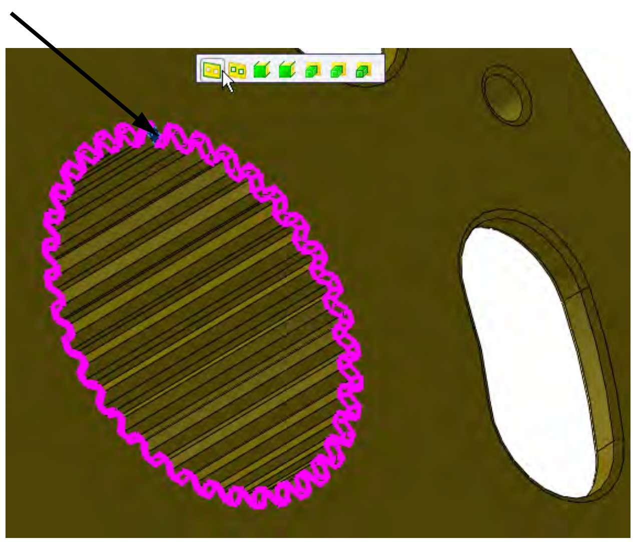
Tip

Fillets can be changed whether they were created with the current FilletXpert PropertyManager or not.

4 Change.
Click the **Change** tab and select the fillet at the base of the circular boss as shown. Set the radius value to **5mm** and click **Resize**.



5 Remove.
Select this set of fillets by selecting the indicated face with **Right Loop** and click **Remove**. Click **OK**.



6 Save and close the model.

FilletXpert Corners

Corner faces generated by fillets can be modified to alternative blends using the **Corner** tab of the **FilletXpert**.

Tip

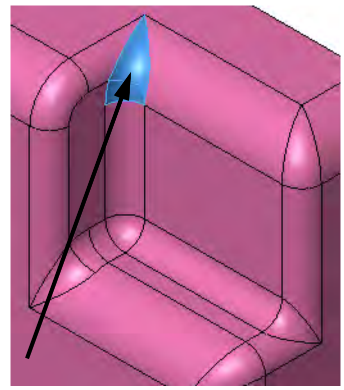
If you try to select a corner that is unsuitable, a message will appear telling you that you must select a corner that has three constant radius fillets of mixed convexity meeting at one vertex.

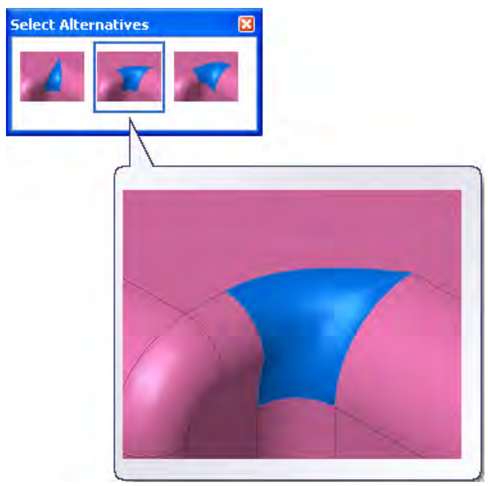
---

1 Open the part Corners.

2 Select face.
Click **Fillet**, **FilletXpert** and **Corner**. Select the face as shown.

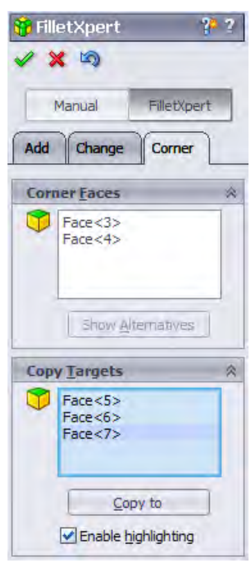
Click **Show Alternatives** and select the alternative shown.

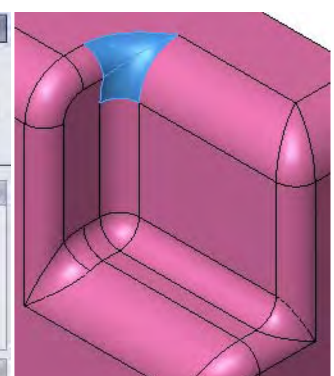




3 Copy to.
The **Copy to** option allows you to apply the same corner alternative to other similar corners.

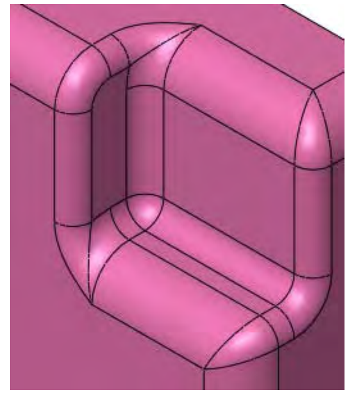
Click the fillet corner that was just changed. Click **Enable Highlighting** and click in the **Copy Targets** selection box. Similar corners highlight. Select them and click **Copy to**.





4 Results.
Click **OK**.

5 Save and close the model.



DraftXpert

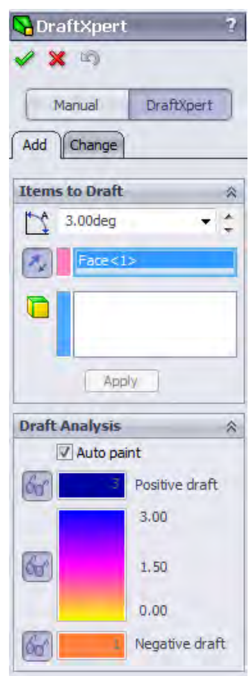
The **DraftXpert** tool is used to add multiple neutral plane drafts of different draft angles, leaving concerns of sequence to the system. It automatically leverages the **FeatureXpert** and **Reorder** to fix potential problems as you create the draft features.

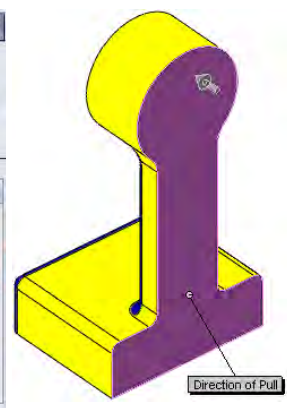
Options within the dialog allow you to **Add** and **Change** draft features.

The **DraftXpert** tool is accessed from the standard **Draft** tool.

1 Open the part DraftXpert.

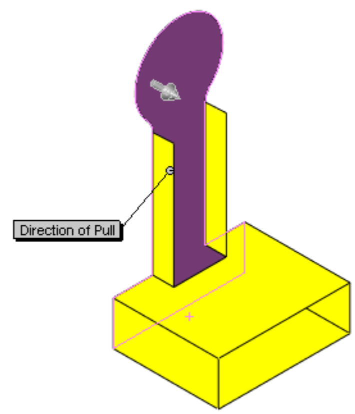
2 Toggle DraftXpert.
Click the **Draft** icon and click the **DraftXpert** button. Click the **Add** tab and click **Auto paint**. Also, select the rear face as the **Neutral Plane**. Reverse the direction if necessary.





Tip

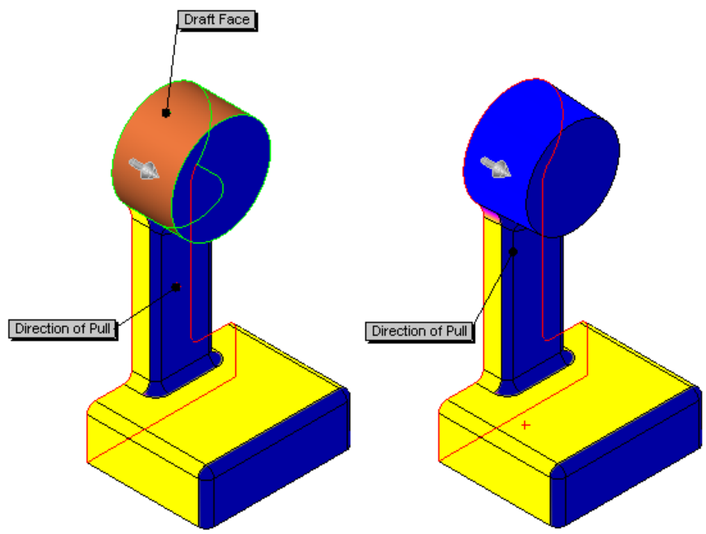
The **Auto paint** option is used to display the current draft, by face, while you work. You can also mouse over a face to get a draft value. Use the **Show/Hide Faces** buttons to show/hide groups of faces. The **Positive Draft** faces are hidden in this example.



3 Draft cylindrical face.

Set the draft angle to **3°**. Select the cylindrical face, as shown, to draft.

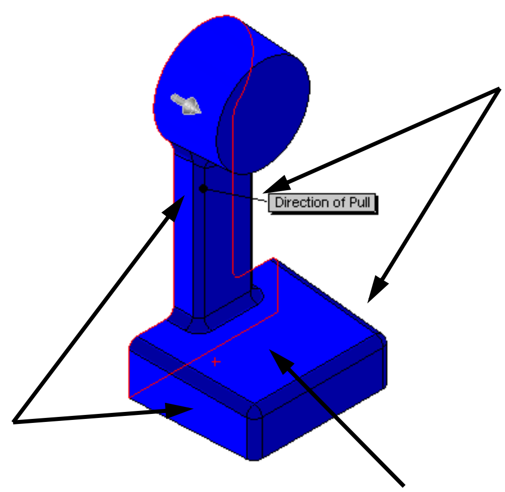
Click **Apply** on the dialog or right click to apply.



4 Planar faces.

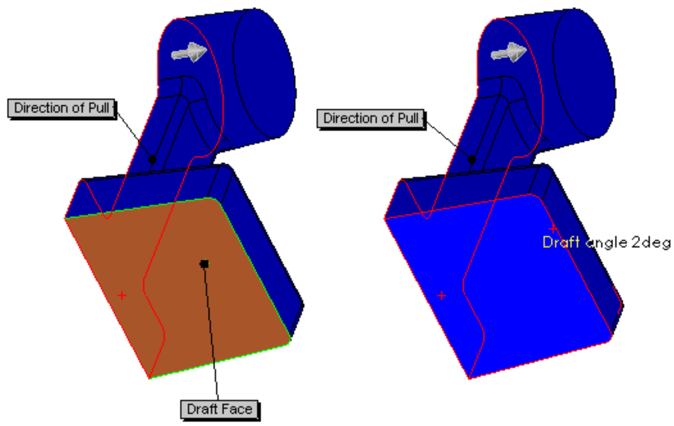
Select the four "vertical" faces and the "horizontal" face of the base.

Click **Apply**.



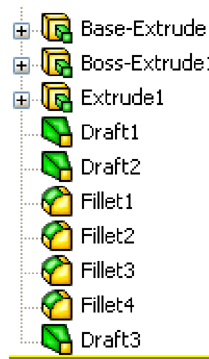
5 Different draft angle.
Set the draft angle to **2°**.

Select the bottom planar face as shown.



Click **OK**.

6 FeatureManager.
Some draft features were created after the extrusions and before fillets. **FilletXpert** and **DraftXpert** have reordered the sequence automatically to prevent rebuild failures.



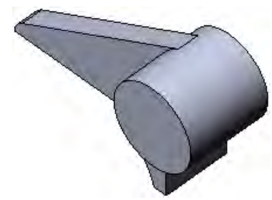
7 Save and close the model.

Exercise 34: Errors1

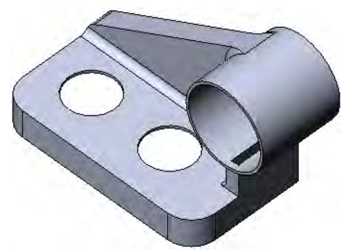
Edit this part using the information and dimensions provided to repair the errors and warnings and complete the part.

This lab reinforces the following skills:

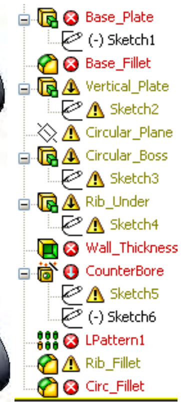
- *What's Wrong Dialog* on page 265.
- *Check Sketch for Feature* on page 270.
- *Reattach Collinear Relations* on page 274.
- *Reattach Dimensions* on page 278.
- *Highlighting Problem Areas* on page 279.



Before

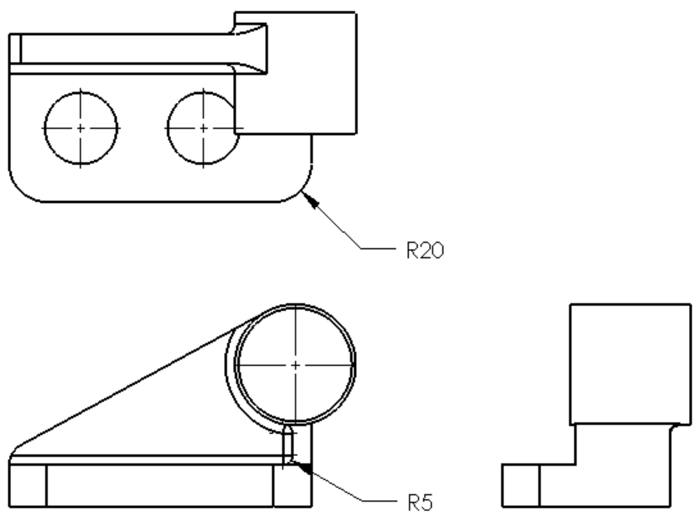


After



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.



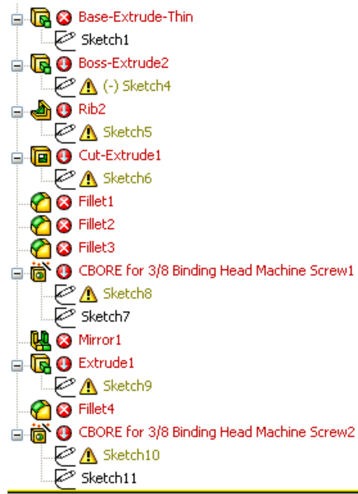
Exercise 35: Errors2

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:

- *What's Wrong Dialog* on page 265.
- *Finding and Repairing Problems* on page 265.
- *Check Sketch for Feature* on page 270.

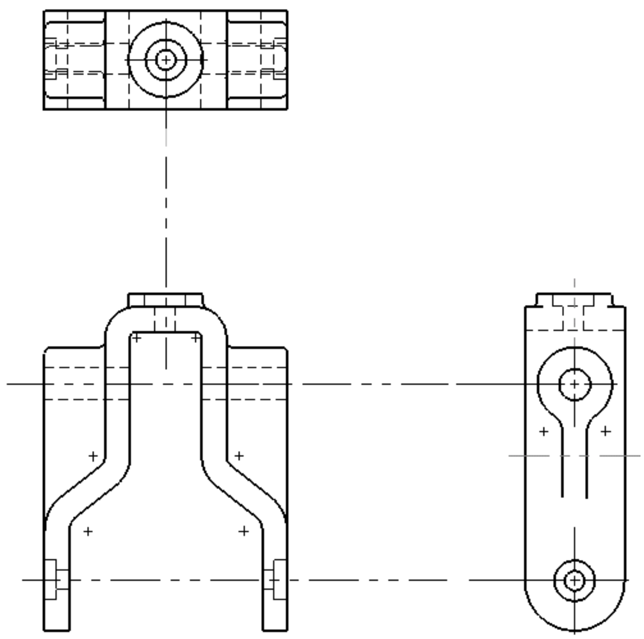
Before



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.

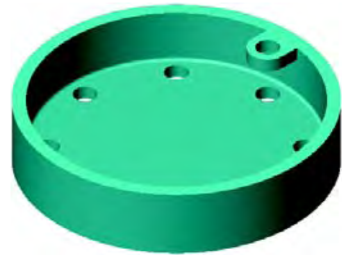


Tip

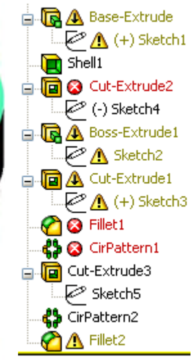
Click **Merge solids** in the `Mirror1` feature. The completed part should be a *single* solid body.

Exercise 36: Errors3

Edit this part using the information and dimensions provided to repair the errors and warnings and complete the part.



Before



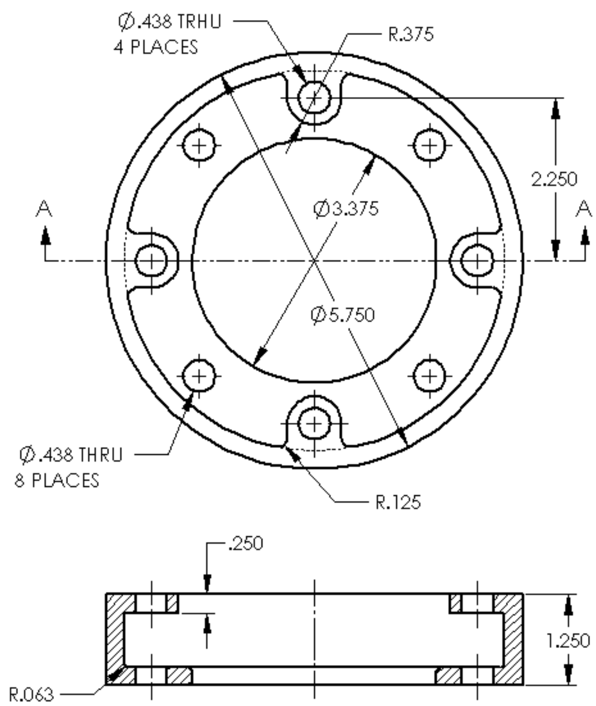
After

This lab reinforces the following skills:

- *What's Wrong Dialog* on page 265.
- *Check Sketch for Feature* on page 270.
- *Repairing Relations Using Display/Delete Relations* on page 275.
- *Reattach Dimensions* on page 278.
- *Highlighting Problem Areas* on page 279.

Procedure

Open the existing part `Errors3` and make several edits to remove the errors and warnings from the part. Use the drawing below as a guide.

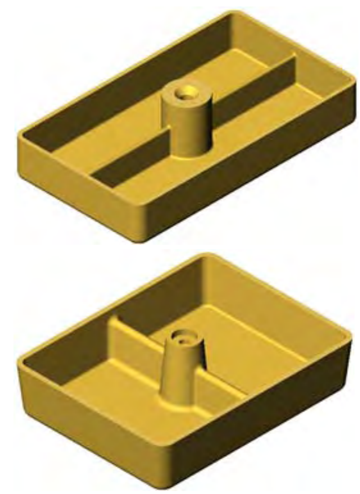


Exercise 37: Adding Draft

Edit this part using the information and dimensions provided. Use editing techniques to maintain the design intent.

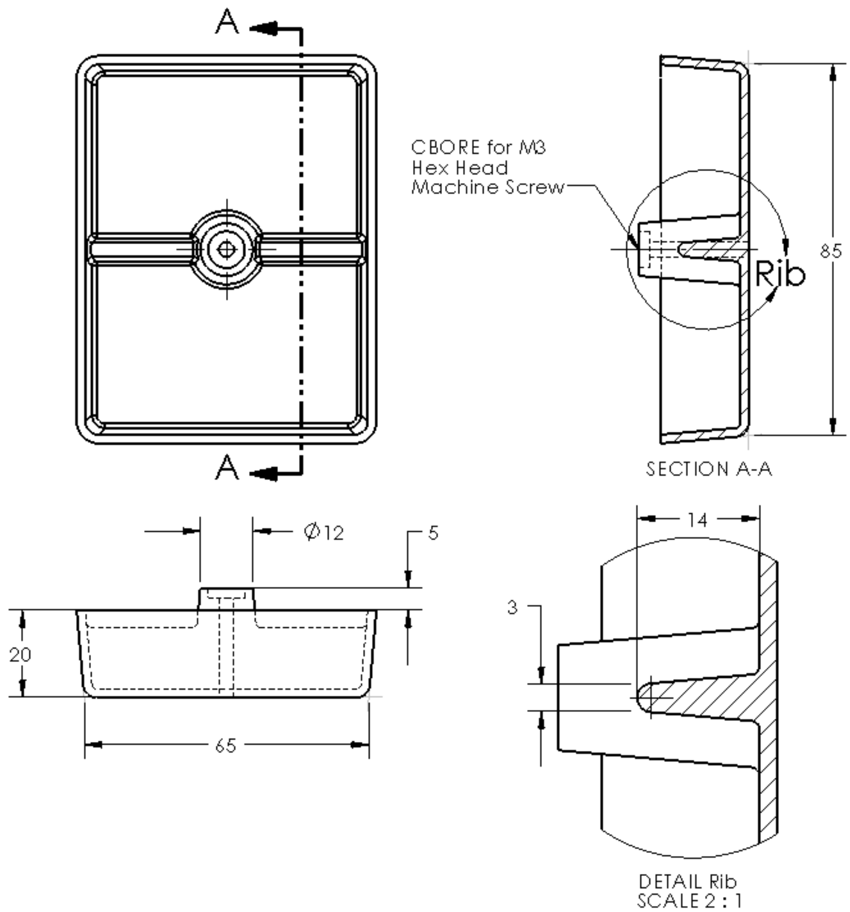
This lab reinforces the following skills:

- *Introducing: Edit Sketch Plane* on page 277.



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.

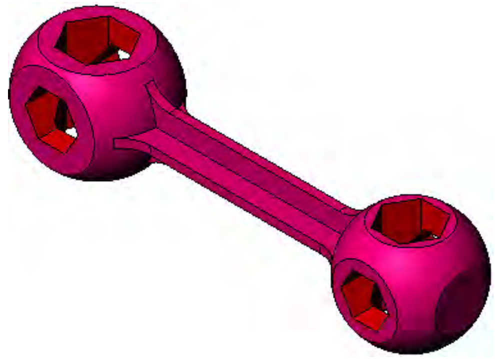


Exercise 38: Copy and Dangling Relations

Complete this part by copying features and making repairs.

This lab uses the following skills:

- *What's Wrong Dialog* on page 265.
- *Reattach Concentric and Coradial Relations* on page 279.



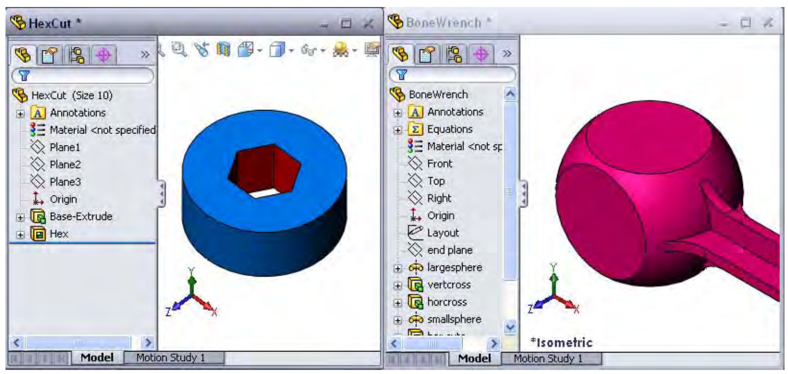
Note

Instant 3D  must be disabled to complete this exercise.

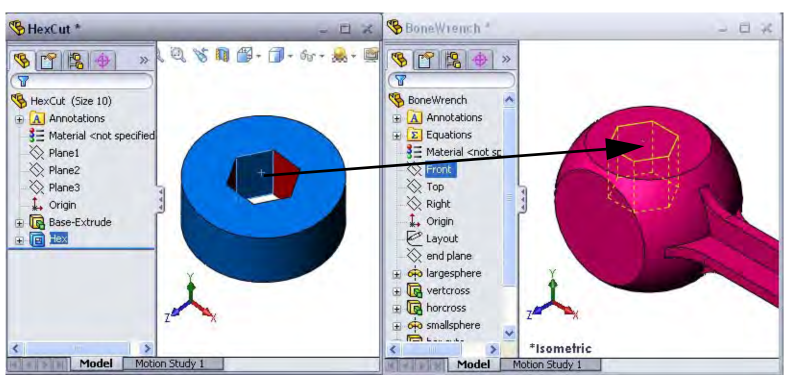
Procedure

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.



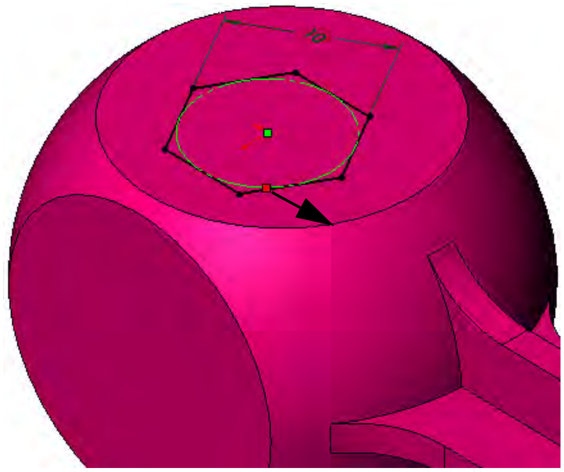
Click the **Dangle** button on the **Copy Confirmation** dialog.

3 Repair.

Edit the sketch with an error and select the inner construction circle.

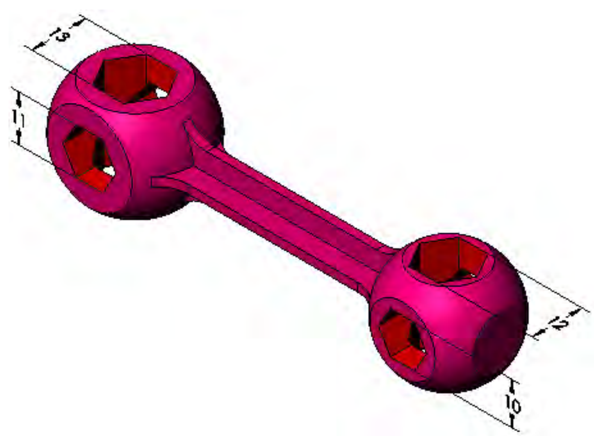
Drag and drop the red marker to an appropriate replacement reference.

Size the hexagon as shown.



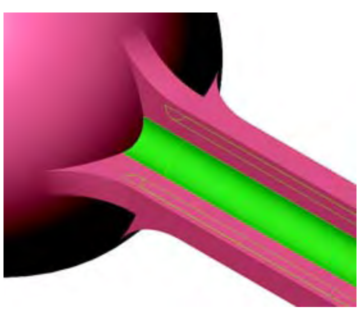
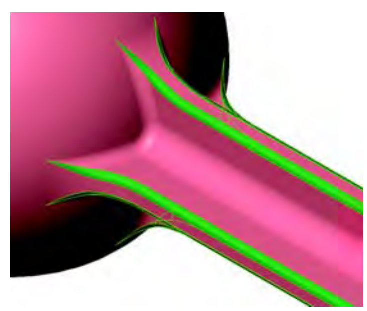
Locations

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
		

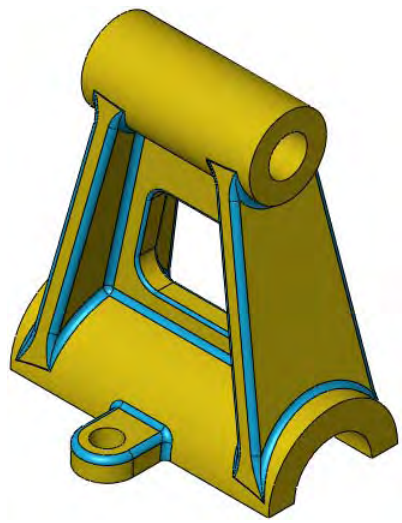
5 Save the parts and close them.

Exercise 39: Using the FilletXpert 1

Complete this part by using both the FeatureXpert and FilletXpert.

This lab uses the following skills:

- *Box Selection* on page 269.
- *FilletXpert* on page 281.

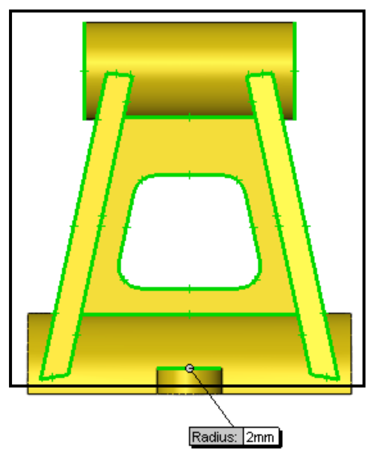


Procedure

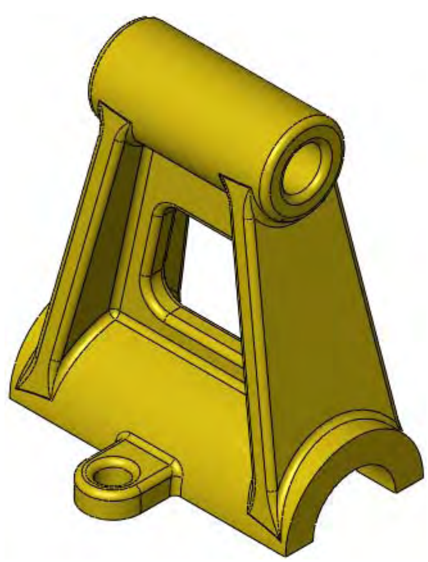
Open an existing part.

1 Open FilletXpert_Lab_1.

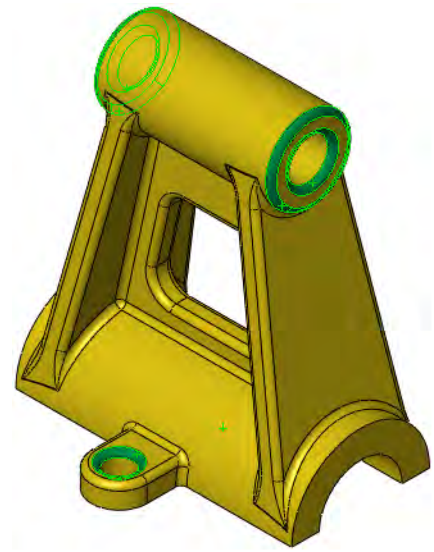
2 Fillet.
Click the **Fillet** tool and the **Manual** button. Set the **Radius** value to **2mm** and **Box Select** the edges as shown. Click **OK**.



3 FeatureXpert.
The What's Wrong dialog explains that the fillets cannot be created. Click the **FeatureXpert** button. The fillets are created.

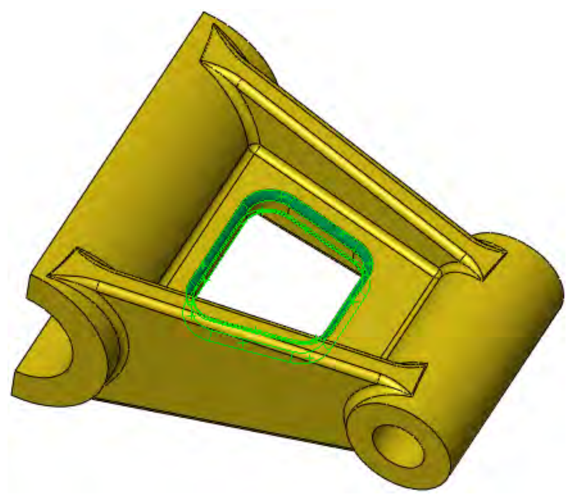


4 Remove.
Use FilletXpert to remove selected fillet faces as shown.



5 Resize.
Use FilletXpert to resize selected fillet faces to **1mm** as shown.

6 Save and close.

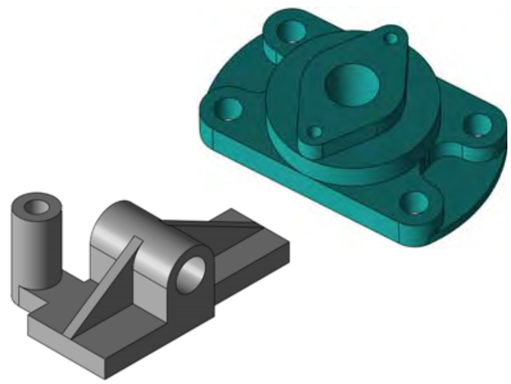


Exercise 40: Using the FilletXpert 2

Complete these parts by using the FilletXpert.

This lab uses the following skills:

- *FilletXpert* on page 281.



Procedure

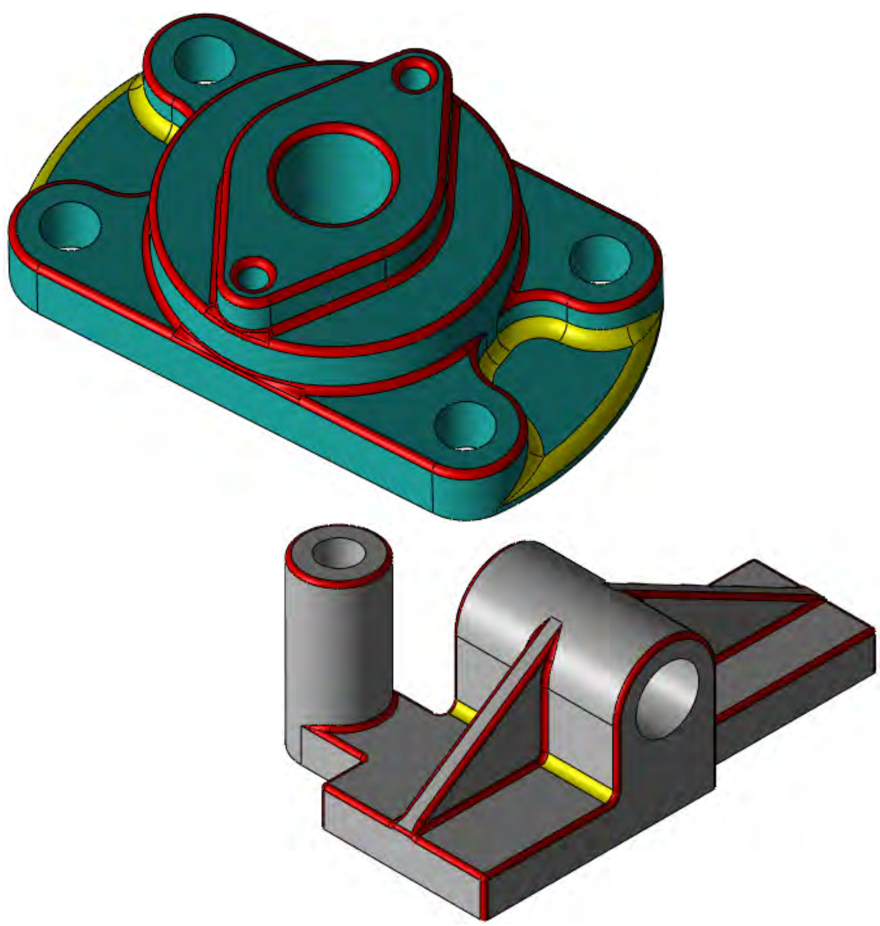
Open existing parts.

1 **Open FilletXpert_Lab_2 and FilletXpert_Lab_3.**

2 **FilletXpert.**
Add the fillets shown with the FilletXpert using this code: red = **1mm** and yellow = **2mm**.

Tip

Sometimes it is easier to fillet all the edges then remove and edit fillets later.



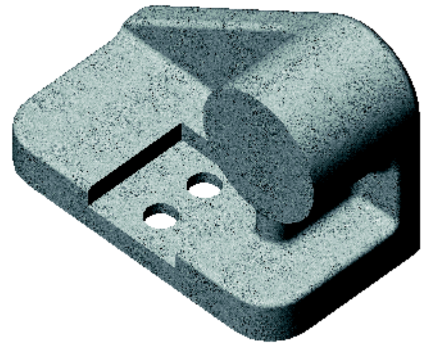
Lesson 9
Editing: Design Changes

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

- Understand how modeling techniques influence the ability to modify a part.
- Utilize all the available tools to edit and make changes to a part.
- Use Sketch Contours to define the shape of a feature.
- Edit with instant 3D tools.

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.

- **Information from a model**
  Many of the most commonly used editing commands: Edit Sketch, Edit Feature, Edit sketch plane, Reorder, Rollback and Change dimension value are used here.
- **Edit the model**
  Use the editing tools 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.

Design Changes

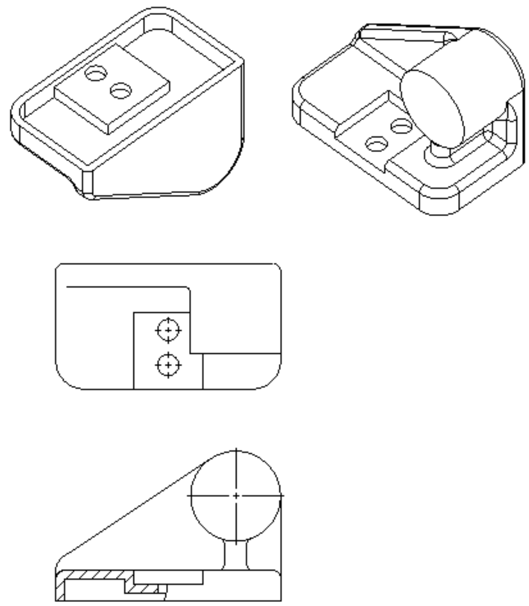
Some changes have to be made to the model. Some will change the structure of it, others only dimension values. Making design changes to a model can be as simple as changing the value of a dimension and as difficult as removing external references. This section steps through a series of changes to a model. The focus is on editing features rather than deleting and reinserting them. Editing enables you to maintain references to drawings, assemblies or other parts that would be lost if you deleted the feature.

Procedure

We will edit a part that was repaired in a manner similar to 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.
- The circular boss is flush with the vertical plate.
- Only the base is shelled.

1 Open the part named `Editing_Design_Changes`.
This part was built with numerous errors and repaired.

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.

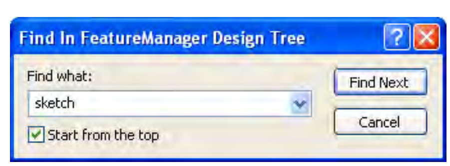
Introducing: Go To

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...**.

2 Go To.
Right-click the top level feature and select **Go To...**. Type the partial name `sketch` and click **Start from the top**.



3 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.

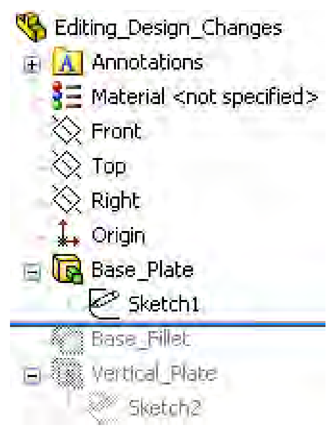
Tip

You can close all expanded features by right-clicking in the FeatureManager and choosing **Collapse Items** or pressing **Shift+C**.

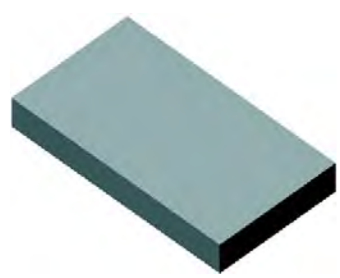
Note

The **FeatureManager Search Box** can also be used to search for text strings.

4 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.

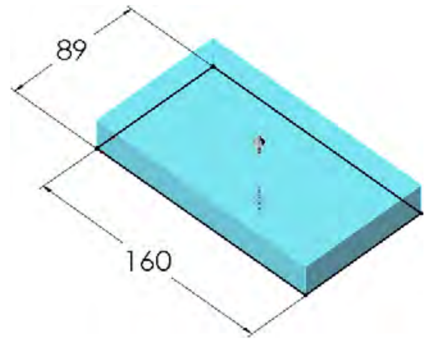


5 Feature `Base_Plate`.
The `Base_Plate` was created from a rectangle and extruded. To investigate this further, use **Edit Feature** on the feature.



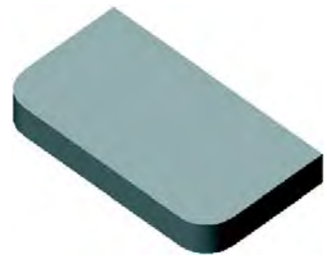
6 Edit Feature.
The graphics show the sketch geometry and the preview. **Cancel** the dialog.

Roll forward one feature by dragging the marker or moving it down with the arrow key.

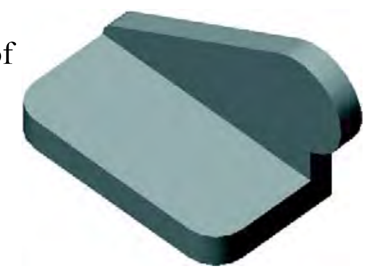


7 Feature Base_Fillet.
Fillets of equal radius are added to the front corners in this feature.

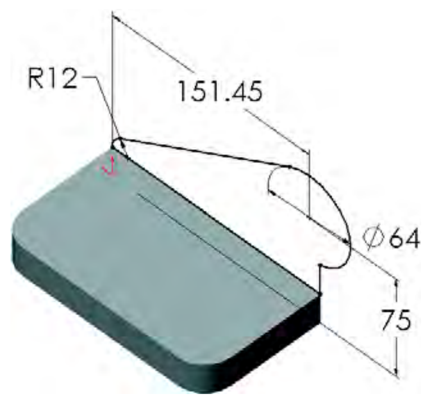
Roll forward to a position just after the Sketch2 feature.



8 Feature Vertical_Plate.
This feature was sketched on the rear face of the model and extruded towards the front.



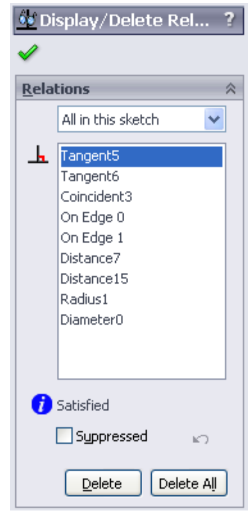
9 Edit Sketch.
Edit the sketch of the feature Vertical_Plate to see the geometry and its connections points.



10 Display/Delete Relations.

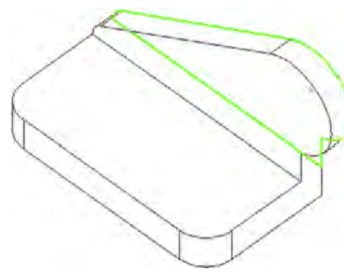
Click **Display/Delete Relations**. Set the **Filter** to **All in this sketch** and click individual relations in the list to explore all geometric relations on the sketch entities. The relations will explain how entities are attached to each other and to the rest of the model.

Close the dialog and close the sketch without making any changes.



11 Sketch geometry.

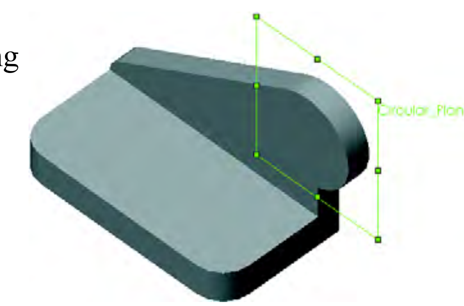
To see the sketch geometry more clearly, right-click Sketch2, and select **Show**. The sketch icon appears in color when it is being shown. Using **Hidden Lines Removed**, the position of the sketch is clear.



Roll forward to a position just after the Circular_Plane feature.

12 Circular_Plane.

The plane was created for sketching the next feature, a circular boss. It lies behind Sketch2.



Dependencies

Dependencies are relationships between features in the FeatureManager design tree. This information is important when editing, deleting or reordering features.

Parents - Features that the target feature depends upon.

Children - Features which are dependent upon the target feature.

Introducing: Parent/ Child Relationships

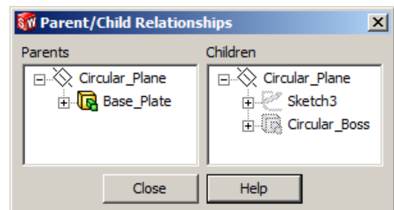
Parent/Child is used to display the dependencies between features. It is used to display the parents and children of a feature.

Where to Find It

- From the right-mouse button over a feature click **Parent/Child**.

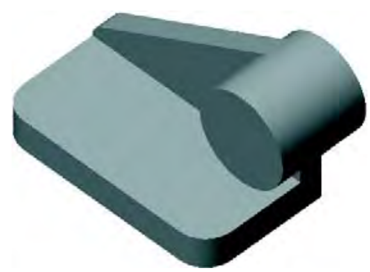
---

13 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.



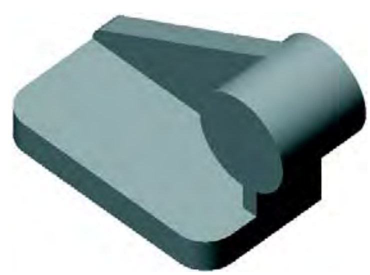
Click **Close** and roll forward.

14 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.

15 Feature Rib_Under.
This feature was sketched as a rectangle and extruded up into the Circular_Boss.

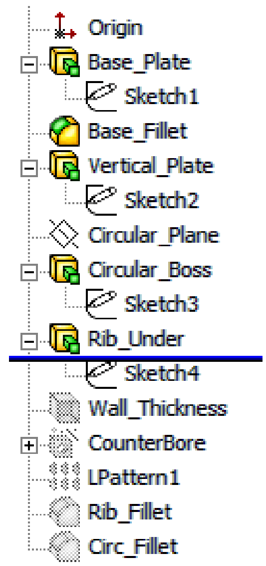


Rollback to a Sketch

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.

16 Rollback to Sketch4.

Move the rollback bar to a position between the Rib_Under feature and its sketch Sketch4. The message appears:



You have chosen to rollback to Sketch4 which is absorbed in feature Rib_Under. The following features will be temporarily unabsorbed for editing purposes: Sketch4

Click **OK**.

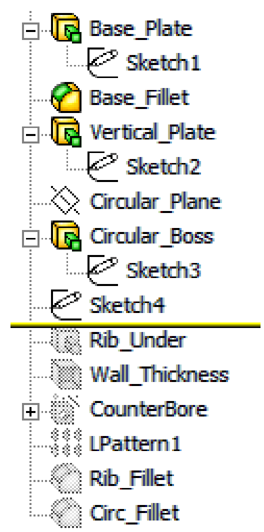
Tip

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*.

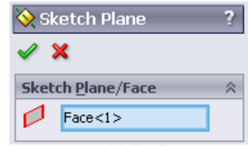
17 Roll Forward.

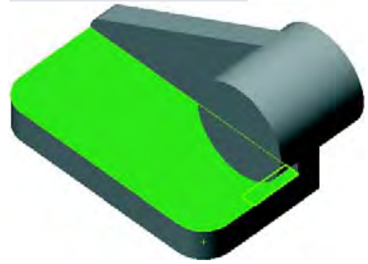
Roll forward to a position after Sketch4.



18 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.

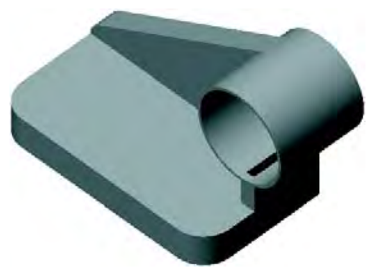
Note

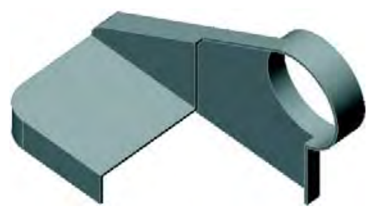
The selection for **Sketch Plane/Face** can be cleared to force the sketch plane reference to dangle. A confirmation message appears in that case:

There is no selection for the sketch plane reference. Select OK if you would like this to be a dangling reference.

19 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.

20 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.

21 Pattern feature.

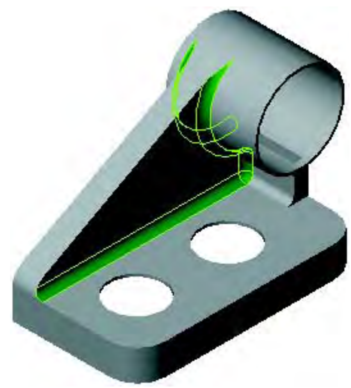
The CounterBore was patterned using a linear pattern, LPattern1.

Roll forward to a position after the Rib_Fillet feature.

22 `Rib_Fillet` feature.
The `Rib_Fillet` feature creates fillets where the `Rib_Under` joins the `Circular_Boss` and `Base_Plate`.

Right-click and select **Roll to End**.

23 `Circ_Fillet` feature.
This feature creates fillets on both sides of the `Vertical_Plate`.

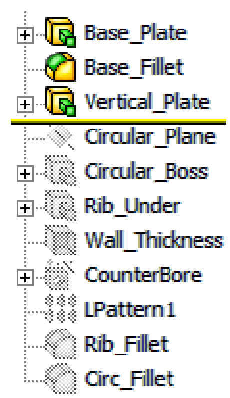


Rebuilding Tools

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.



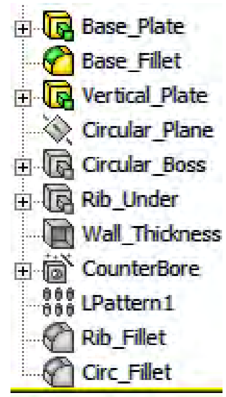
Changes are made to the feature and it is rebuilt. Due to the rollback position, only the features *before* the bar are rebuilt, limiting the scope of the rebuild. The remainder of the part will be rebuilt when the bar is moved.

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 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.



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 edit 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...**.

1 **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 $^1/_4$ 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 is 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 275 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*4 Edit Feature.* on page 311 for an example of changing relations in a feature.

General Tools

In general, there are four tools available to modify features:

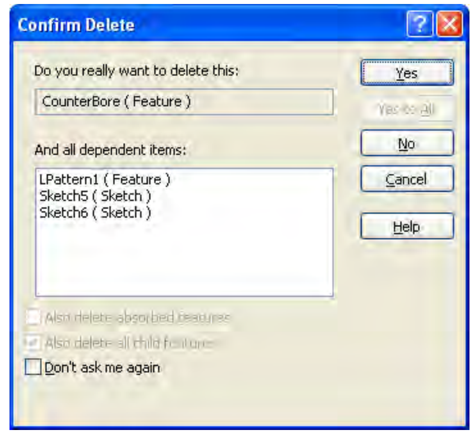
- **Edit Feature** (see *Introducing: Edit Feature* **on page 82**)
- **Edit Sketch** (see *Introducing: Edit Sketch* **on page 81**)
- **Edit Sketch Plane** (see *Introducing: Edit Sketch Plane* **on page 277**)
- **Delete Feature**

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.

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.



Reorder

Reorder allows for changes to the sequence of features in the model. Sequence changes are limited by parent/child relationships that exist. Reorder features by dragging and dropping them onto other features in the FeatureManager design tree. They are placed *after* the feature that they are dropped onto.

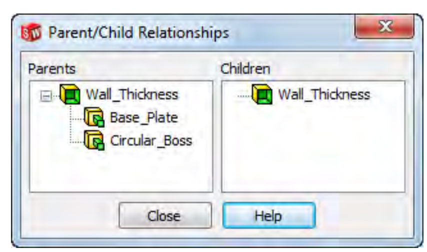
Note

You cannot reorder the target (child) feature before the parent feature.

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. If you try to drop the feature, this message appears:

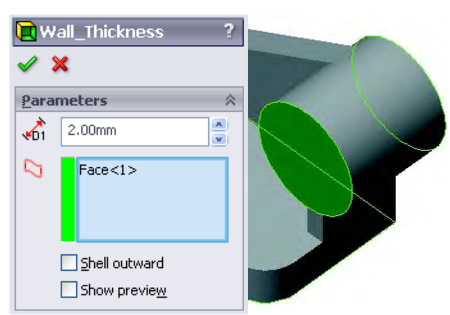
Cannot reorder. Change would put child feature before parent feature.

The Circular_Boss references need to be removed in order for us to be able to reorder the feature.



4 Edit Feature.
Right-click the Wall_Thickness feature and select **Edit Feature**.

Select both of the highlighted circular faces. The **Faces to Remove** selection list will show only a single face.



Note

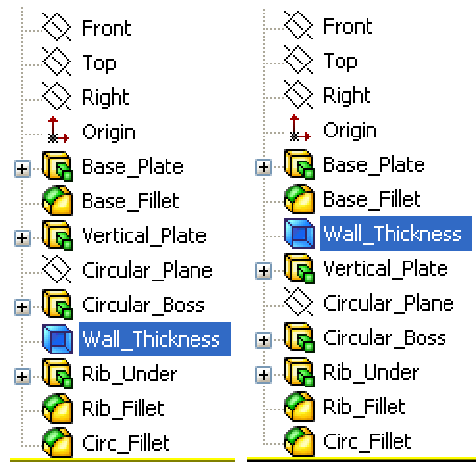
When you reselect an already selected face, it acts like a toggle, deselecting it.

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>.

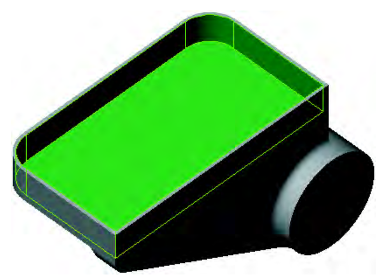
5 Changes to dependencies.
Editing the Wall_Thickness feature causes a change in the dependencies. The parent is now in only one feature, the Base_Plate.

The feature can now be reordered.

6 Reorder.
Drag the Wall_Thickness feature and reorder it by dropping it on the Base_Fillet. It is positioned after the Base_Fillet feature.

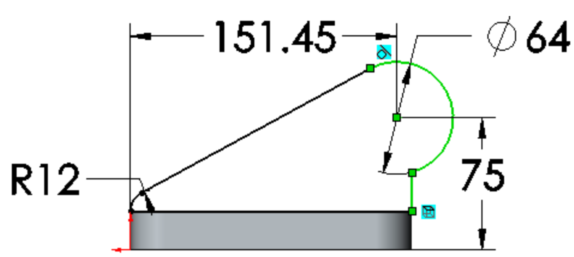


7 Results.
Now the shelling operation affects only the first and second features of the part.



8 Editing the sketch.
Edit the sketch of the Vertical_Plate feature.

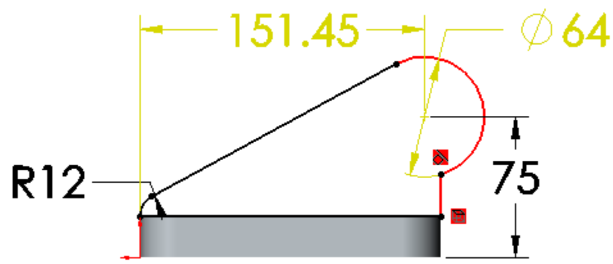
9 Add new relation.
Hold down **Ctrl** and select the rightmost vertical line and the arc. Right-click and select **Make Tangent** to add a **Tangent** relation between the line and the arc.



Overdefined Sketches

If the status of the sketch changes from fully defined to over defined (see *Status of a Sketch* on page 38), a diagnostic tool appears. This tool can be used to repair the sketch. Other unfavorable states can be repaired as well.

10 Over defined.
The addition of the relation triggers an error in the sketch. The sketch is now over defined.



SketchXpert

The **SketchXpert** option is used to automate the repair of Over Defined, No Solution Found, or Invalid Solution Found conditions in the sketch. The SketchXpert is based on SolidWorks SWIFT technology.

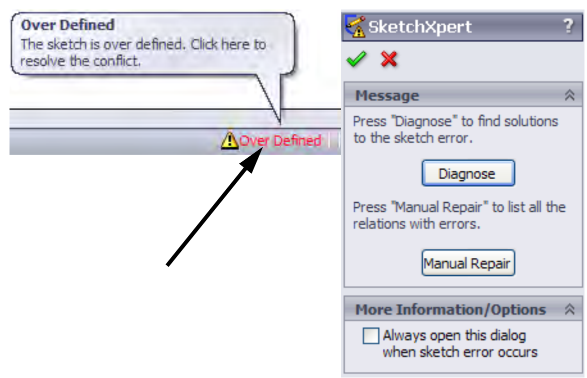
Note

General part editing and repairs were discussed in *Lesson 8: Editing: Repairs*.

Where to Find It

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

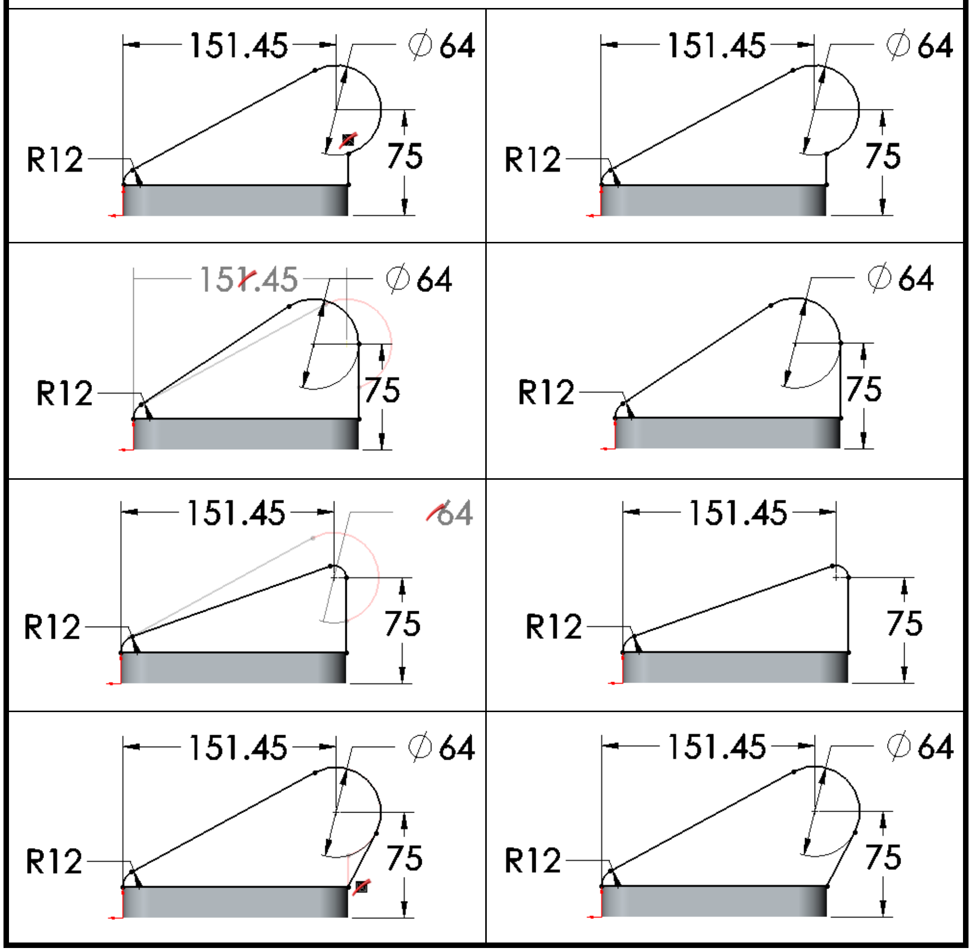
11 Over Defined.
When the sketch becomes Over Defined, a message pops up from the lower right corner of the screen. Click on the text **Over Defined**.



12 Diagnose.

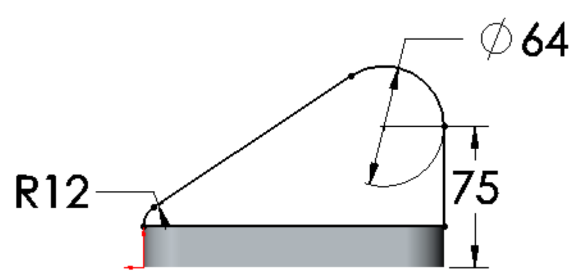
Click **Diagnose** and click the >> button to walk through the solutions. Each solution represents a possible solution. Each set uses a different combination of relations and dimensions. Relations or dimensions that will be removed by selecting that set are marked with a red line through them. They are also listed in the More Information/Options group in the PropertyManager.

Possible Solutions 1-4



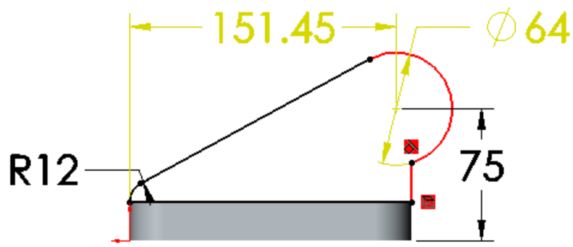
13 Select.

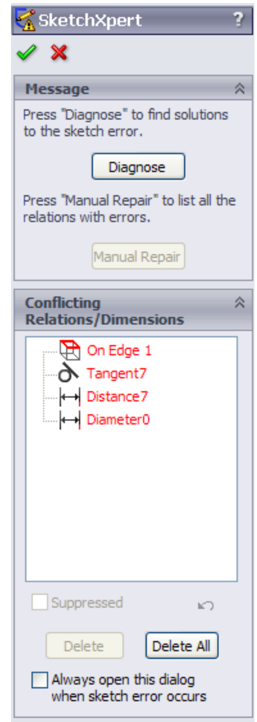
Accept the solution that deletes the horizontal linear dimension.



Manual Repairs

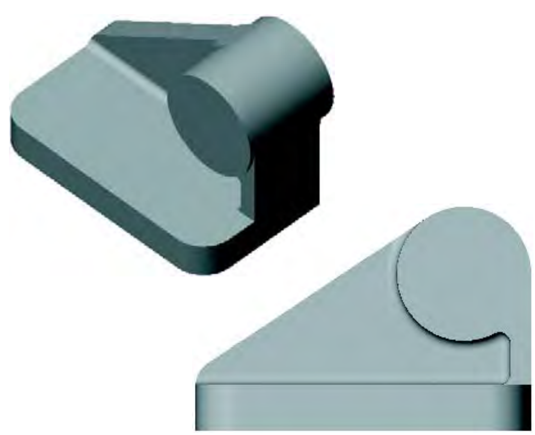
The **Manual Repair** option can also be used to resolve the over defined state. Using this option, the conflicting relation or dimension is selected and deleted.



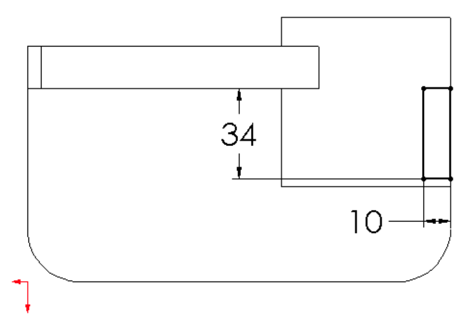


14 Exit the sketch.

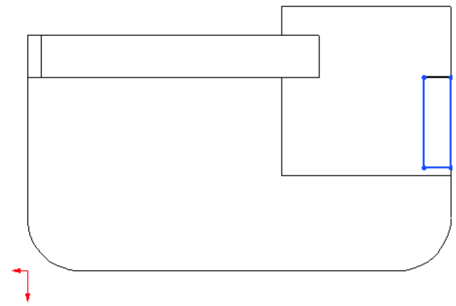
15 Resulting model.
This moves the `Circular_Boss` so that its cylindrical face is tangent to the outer edge of the `Base_Plate`. The fillets update to the new positions.



16 Edit the `Rib_Under` sketch.
The `Rib_Under` sketch is still tied to its original relations, the outer edge of the `Base_Plate`.



17 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.

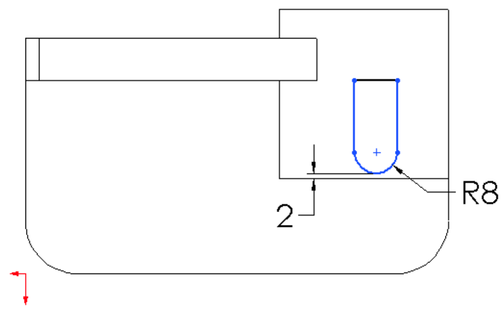


Select and remove these relations using the **Delete** button:

- **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.

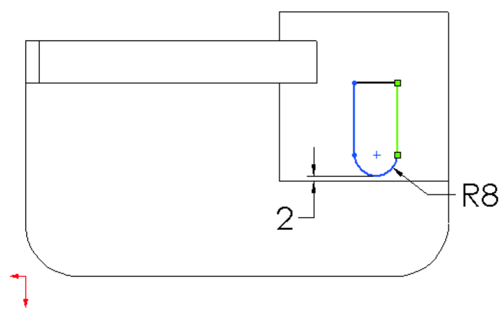
18 New geometry.
Delete the bottom line of the rectangle and add a tangent arc. Dimension the sketch as shown.



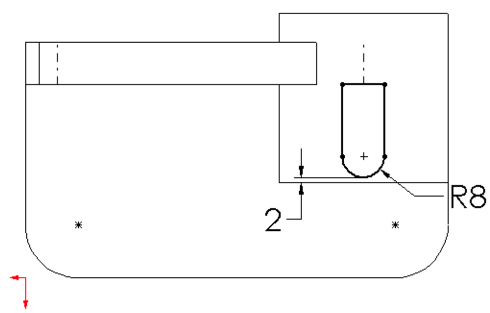
Tip

Shift+select the arc when creating the **2mm** dimension.

19 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.

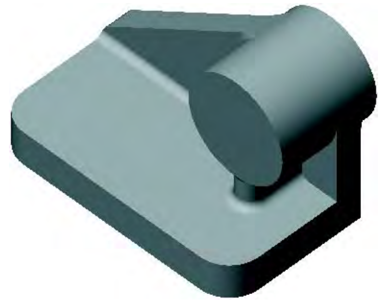


20 Temporary graphics.
Turn on display of **Temporary Axes** and relate the center of the arc to the temporary axis. This will center the rib on the circular boss. Close the sketch.



21 Result.

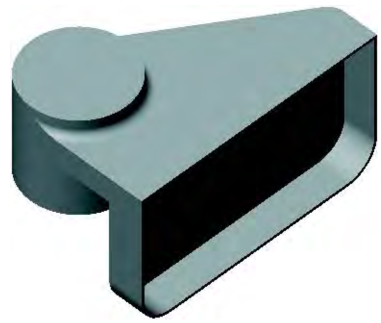
The Rib_Under feature is now centered under the Circular_Boss. It has a rounded front edge and is also inside the edge of the boss by a small amount.



22 Edit Sketch Plane.

Expand the listing of the Circular_Boss feature. Right-click the sketch and select **Edit Sketch Plane** from the shortcut menu.

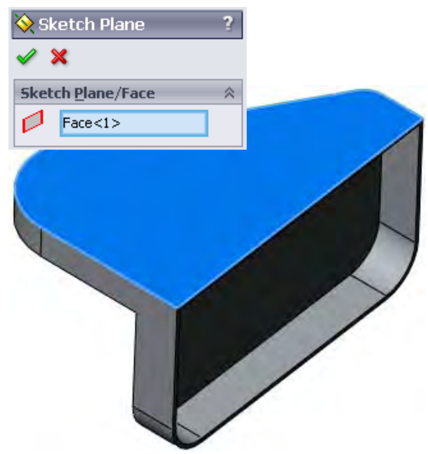
You do not have to edit the sketch.



23 Face or plane selection.

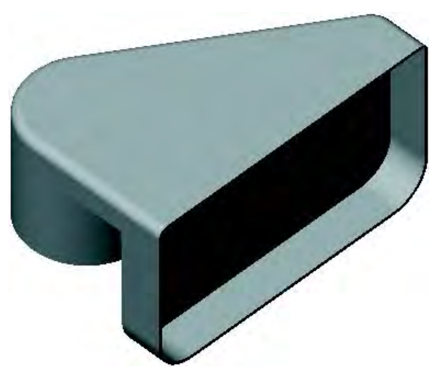
The current plane used in the sketch is highlighted. You can now choose a new sketch plane.

Select the rear face of the model and click **OK**.



24 Edited sketch plane.

The Circular_Boss feature has been edited. The sketch now references a model face rather than a plane.

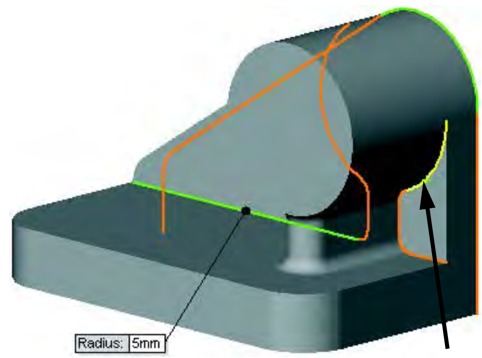


25 Delete the plane.

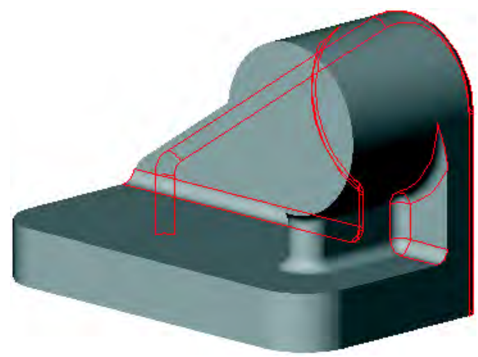
Check the **Parent/Child Relationships** of the plane. The Circular_Plane now has no children.

Delete the plane.

26 Edit Feature.
Edit the Circ_Fillet feature.
Add the edge shown and click **OK**.



27 Result.
The additional edge is filleted as part of the Circ_Fillet feature.

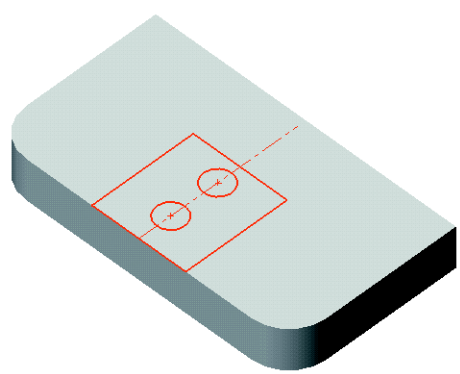


28 Save and Close.
An existing part will be used for the reminder of the case study.

29 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.

30 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.

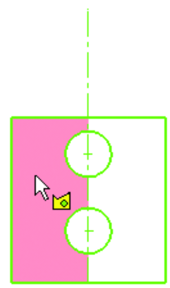
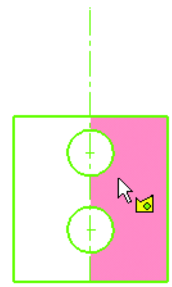
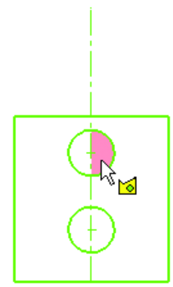
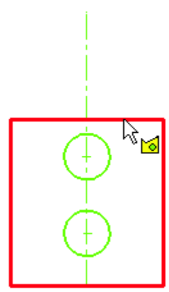
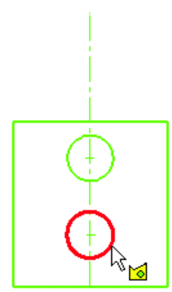
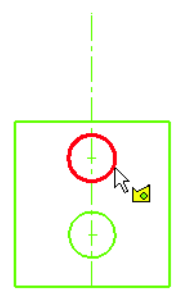
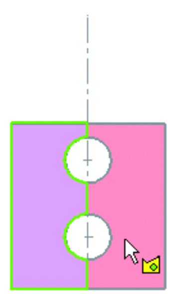
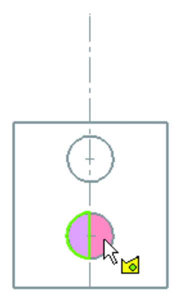
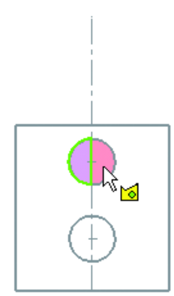
Another advantage of this method is that the sketch can be reused, creating separate features from different portions of the sketch.

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 to 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 .

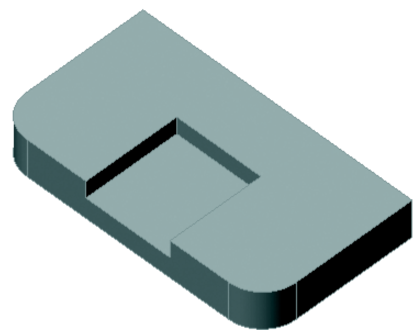
---

31 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.



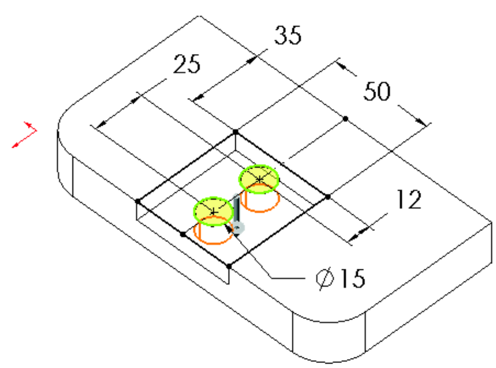
Shared Sketches

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.

32 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.

33 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.

34 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.

35 Change wall thickness.
Change the wall thickness to **6mm** and rebuild to complete the model.

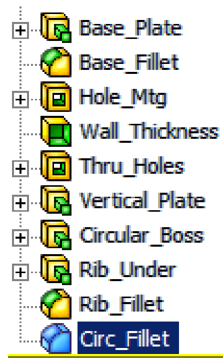
Copying Fillets

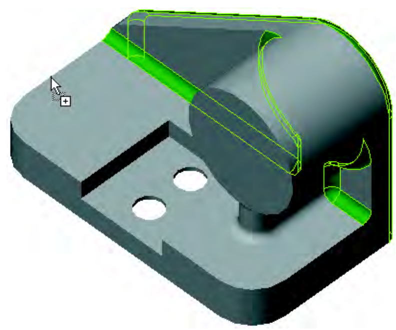
A quick and easy way to create a new fillet is to copy it from an existing feature. The new fillet is the same type and size but unrelated to the original.

36 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.





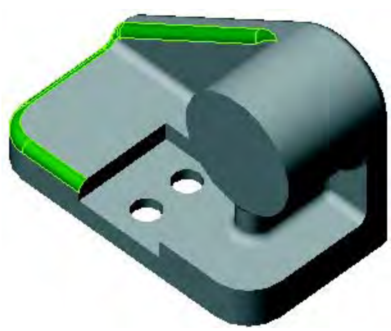
Tip

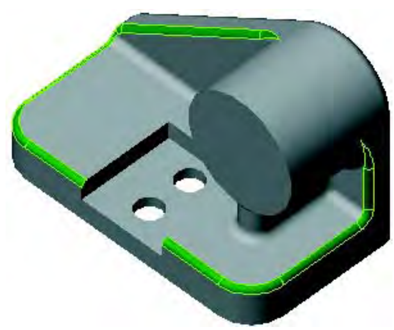
Chamfers can be copied using the same procedure.

37 New fillet feature.

A new fillet feature is created on the edge.

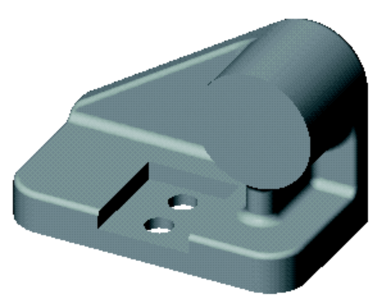
Edit the fillet and add the edge on the opposite side. Change the radius value to **3mm**.





38 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. Planes or planar faces can be used.

Where to Find It

- Click **Section View** on the View toolbar.
- Or, click **View, Display, Section View**.

Note

Drawing view sections can also be created by clicking **OK** and saving a **Drawing Annotation View**. They are added to the drawing using the View Palette.

---

39 Select Face.

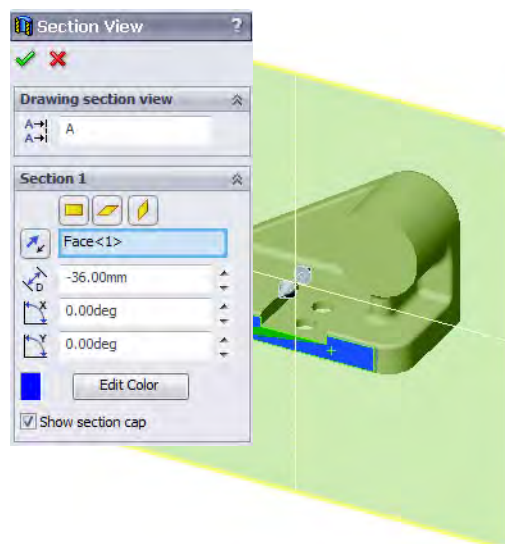
Select the planar face indicated. It will be used to define the section plane.

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`.

40 Section view.

Click **Section View** to use the selected face as the section plane.

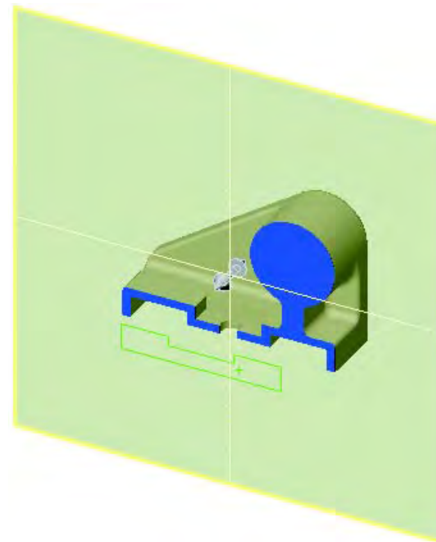


The **Section 1** group box includes options for the **Reference Section Plane**, **Section Direction**, **Offset Distance** and **Angles**.

41 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.

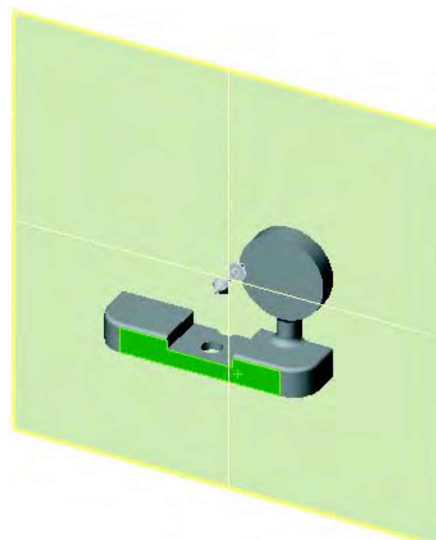


42 Reverse section direction.

Click **Reverse Section Direction**  to reverse the direction of the section.

Click **Cancel** to close the dialog.

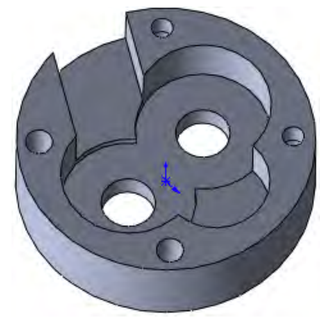
43 Save and close the model.



Editing with Instant 3D

The tools of **Instant 3D** allows you to create geometry quickly and accurately. It includes drag and drop tools with on screen rulers to change dimension values. The tools include:

- Rulers to measure depth.
- One-click value change.

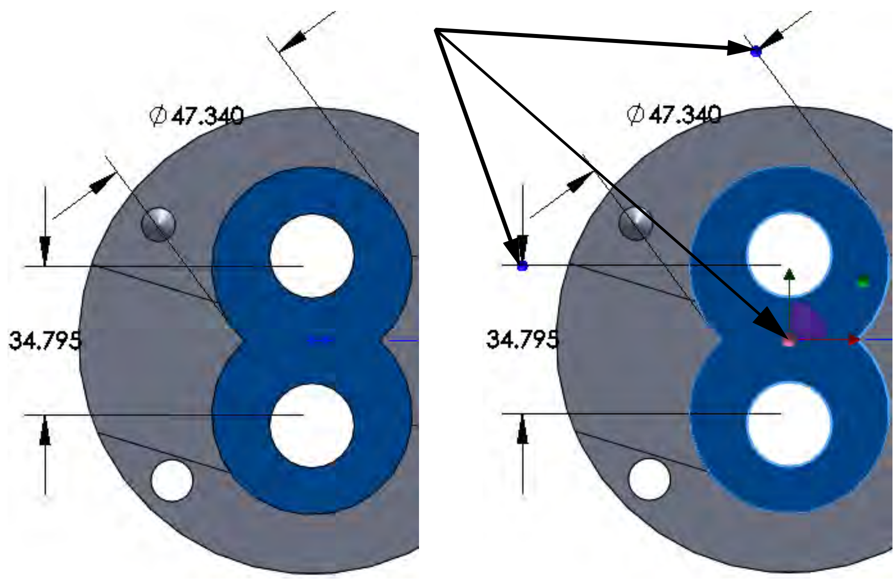


Where to Find It

- Click **Instant3D** from the Features toolbar.

Instant3D Handles

These tools depend on the **Instant3D** icon, which must be toggled on. The **Instant3D** mode displays spherical handles to move and resize geometry by dragging using a ruler.



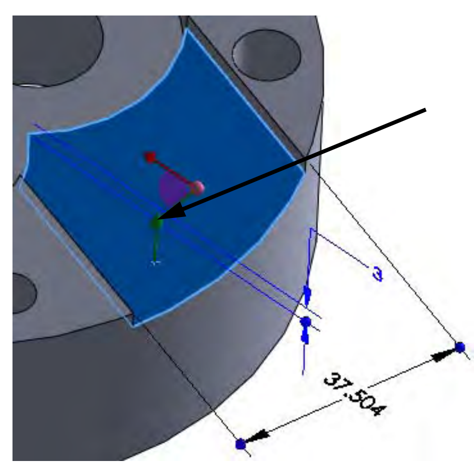
Off On

Dragging Face Geometry

When a face is selected, there are several tools available to edit the geometry of that face and the feature it belongs to.

Face Normals

Face normals appear when faces are selected in Instant3D mode. In some cases, they can be dragged to reposition that face, as shown at right.



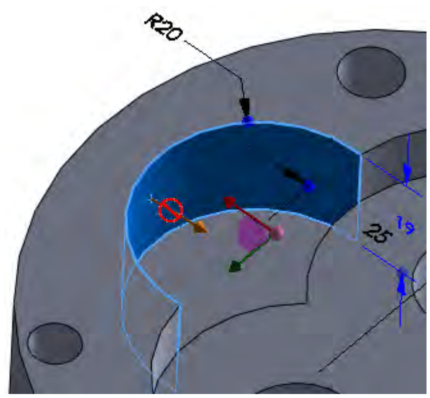
If the face is dragged in the opposite direction of the extrusion, the feature is modified. It can be changed from a boss to a cut, or vice-versa, depending on the direction and distance that it is dragged.

Note

If the face is fully defined, the message appears:

`This object is fully constrained.`

A symbol appears, and the normal cannot be dragged.



Dragging to a Face or Vertex

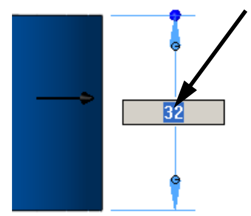
The ruler dimension can be set by a face or vertex selection when a face normal is used. See *Drag to Depth* on page 327 for more information.

Moving the Feature

The 2-axis triad that appears on face selection can be used to move the feature if the feature is under defined. You can also choose to delete the constraints that are holding it in order to move it.

One Click Changes

Instant3D mode also allows for one-click value changes. Click the dimension text and it can be changed directly.



Tip

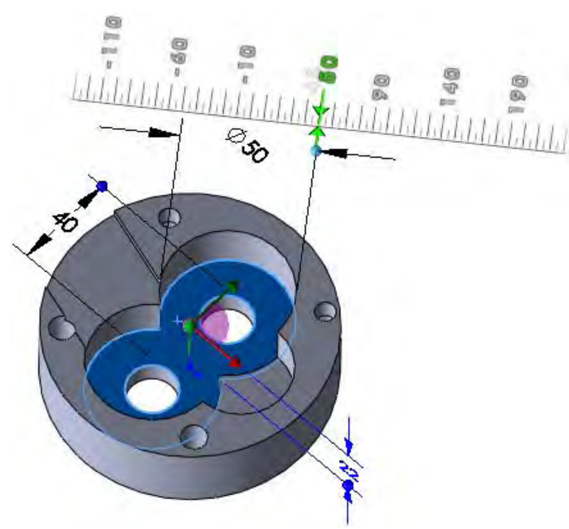
Anytime you are making changes to a model, using this or other methods, errors and warnings may appear and require repairs. Refer to *Editing: Repairs* and the previous sections of this lesson for reference.

---

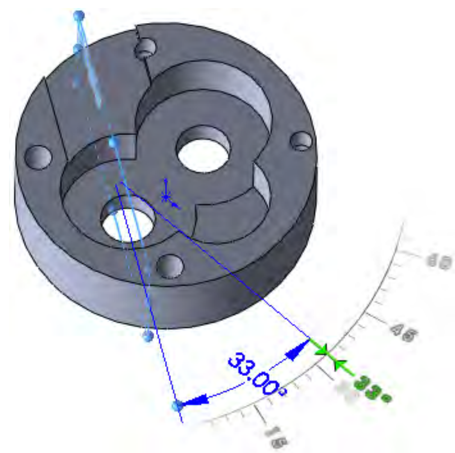
1 **Open the part** `Instant`**.**

2 **Drag dimensions.**
Click the `figure_eight` feature and set these values using drag or direct selection of the dimension as shown.

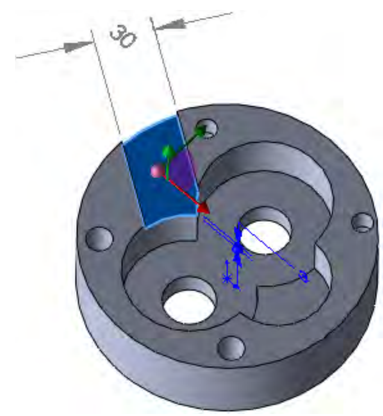
The dimensions are: **40mm**, **50mm** and **22mm**.



3 Drag angle.
Click the angle plane and drag to **33°** as shown.



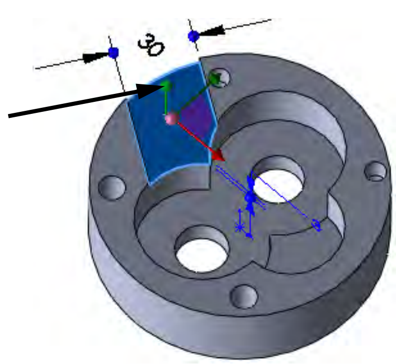
4 Width.
Click the dimension and use One Click Change to make it **30mm** as shown.

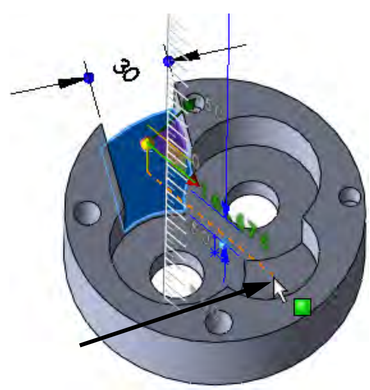


Drag to Depth

Using the **Alt** key, the depth of an extrusion can be set to the same value as an existing depth defined by a face or vertex selection.

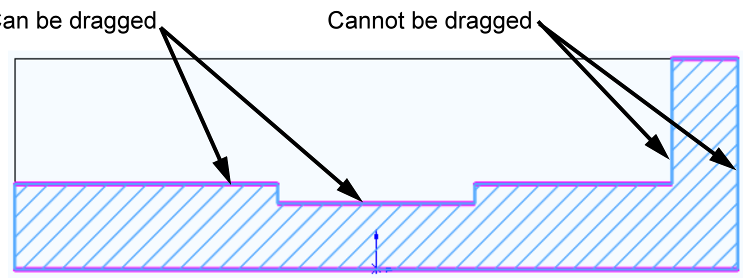
5 Drag using Alt.
Click the face of the out feature as shown and drag the face normal. Press and hold the **Alt** key while moving the cursor to the face of the well feature and drop it as shown. The depths of both features are set to the same depth value **19mm.**





Live Section Plane

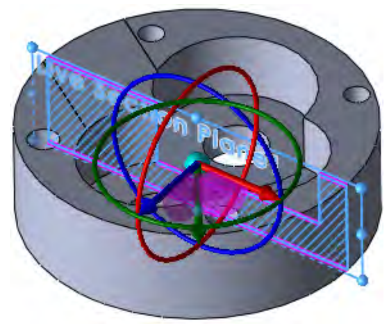
The **Live Section Plane** is used to dynamically section the model with a plane. The thickened edges of the section can be selected and dragged with rulers. The unthickened edges are not editable.



Where to Find It

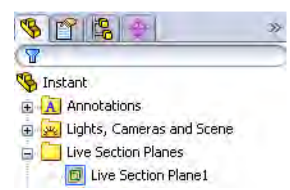
- Right-click a plane or planar face and select **Live Section Plane**.

6 Create live section plane.
Right-click the `Front` plane and select **Live Section Plane**. The section plane is added to the part with an active triad.



Live Section Plane Folder

The **Live Section Plane** feature is stored as `Live Section Plane1` under the `Live Section Planes` folder of the FeatureManager Design Tree.

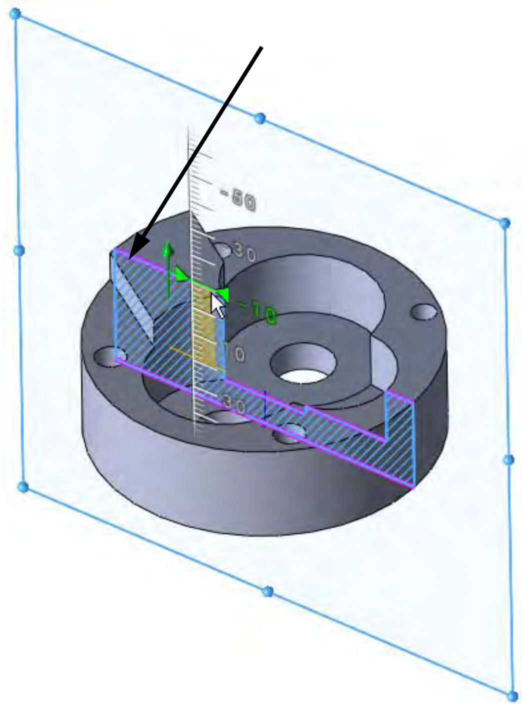


It can be shown/hidden or suppressed/unsuppressed from this position.

7 Drag edge.

Right-click the `Live Section Plane1` feature and select **Fit To Part**.

Drag the edge as shown upward, reversing the direction of the extrusion. The distance value is not important.

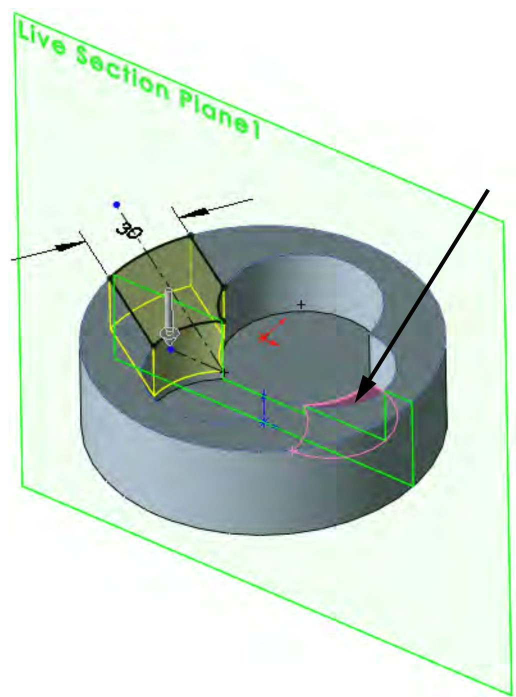


Note

It is clear that although the depth values of the `out` and `well` features were equal in step **5** on page 327, they are not tied together.

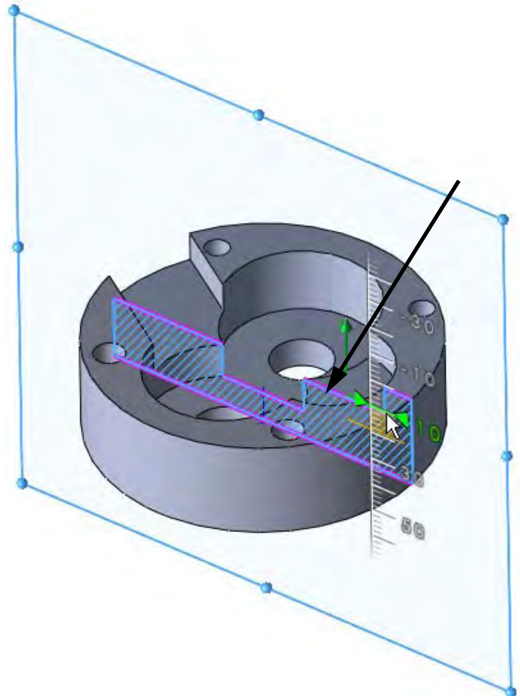
8 Up to surface.

Click **Undo** to negate the change. Right-click the later of the two features, `out`, and select **Edit Feature**. Using **Up To Surface**, select the top face of the `well` as shown and click **OK**.

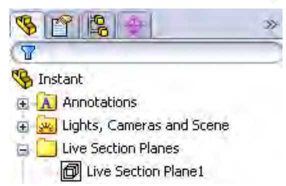


9 Drag another edge.
Drag the edge as shown upward, setting the value at **10mm**.

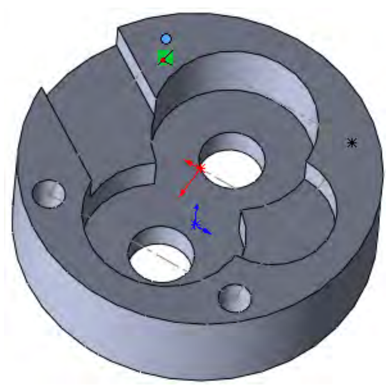
This time the edge of the well feature is dragged, rather than the edge of the out feature. This is due to the relationship that was created in the previous step.



10 Hide live section plane.
Expand the Live Section Planes folder. Right-click the Live Section Plane1 and select **Hide** .

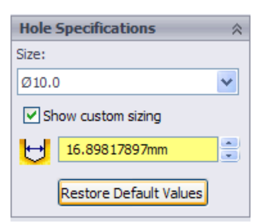


11 Hole repairs.
Show Sketch7 of the Ø10.0 (10) Diameter Hole1 feature. Double-click the Sketch9 of the Ø8.0 (8) Diameter Hole1 feature to edit it. Locate the points on the sketch endpoints as shown by drag and drop.



Tip

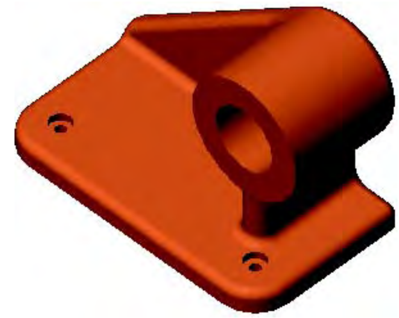
Dragging the geometry of a Hole Wizard feature, except for blind hole depths, should be avoided. These dimensions are standard sizes and should not be changed. If they are inadvertently changed, edit the feature and click **Restore Default Values**.



12 Save and close the model.

Exercise 41: Changes

Make changes to the part created in the previous lesson.



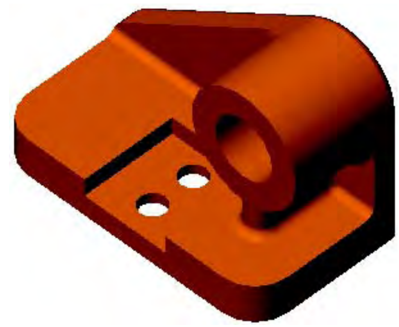
This exercise uses the following skills:

- *Deletions* on page 310.
- *Reorder* on page 311.
- *Copying Fillets* on page 322

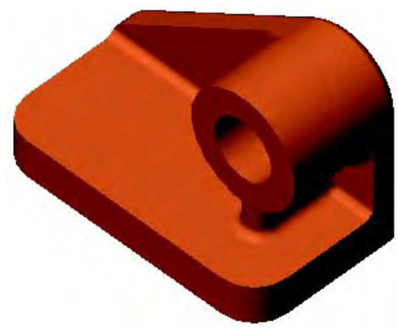
Procedure

Open an existing part.

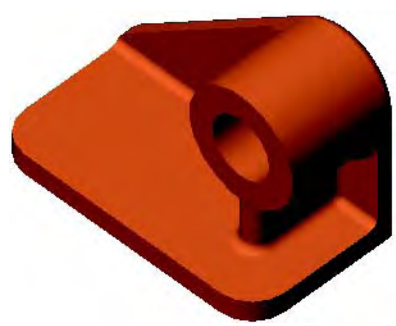
1 **Open the part Changes.**
Several changes and additions will be made to the model.



2 **Delete.**
Delete the mounting holes, cutout and shell (Cut-Extrude1, Wall_Thickness and Cut-Extrude2) and absorbed features from the model.

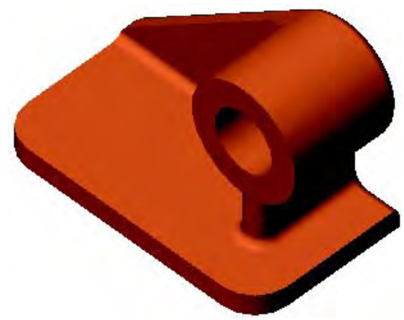


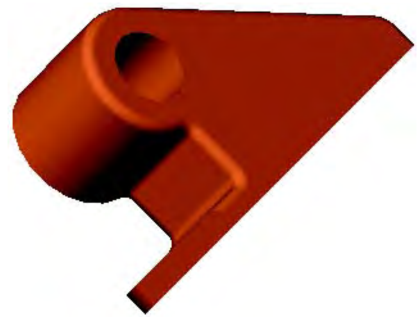
3 **Same thickness.**
Set the thicknesses of the Base_Plate and Vertical_Plate to the same value, **12mm**.



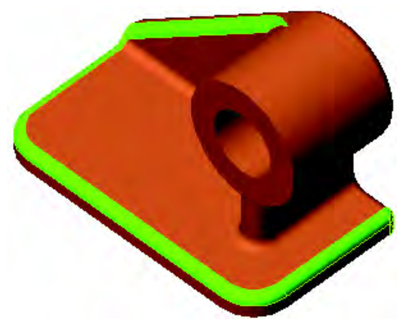
4 Cut.
Remove the portion of the Vert_Plate on the right side of the Circular_Boss and Rib_Under.

Edit, **Rollback** and **Reorder** features where necessary to maintain the filleting.





5 Fillet.
Add another fillet the same radius as the Circ_Fillet.



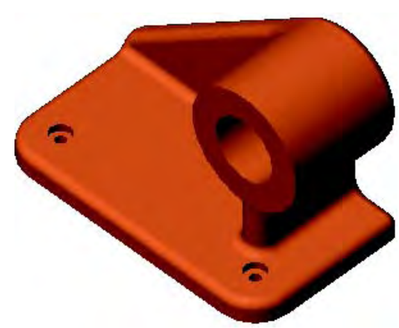
6 Counterbored holes.
Add two counterbored holes of the following size:

ANSI Metric

M6 Hex Cap Screw

Through All

Reorder features where necessary to avoid undercuts.



7 Save and close the part.

Exercise 42: Editing

Edit this part using the information and dimensions provided. Use relations, up to surface end conditions to maintain the design intent.

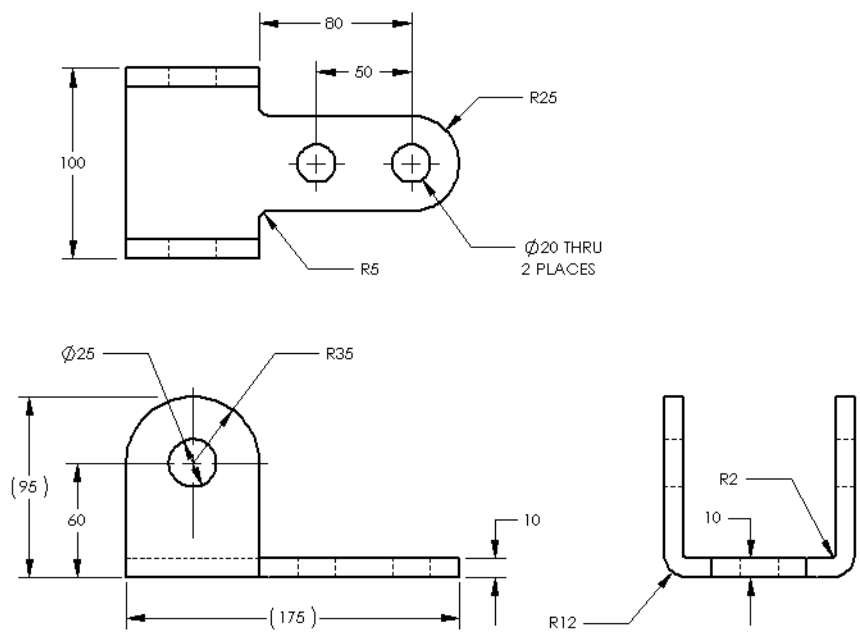
This lab reinforces the following skills:

- *Reorder* on page 311.

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.

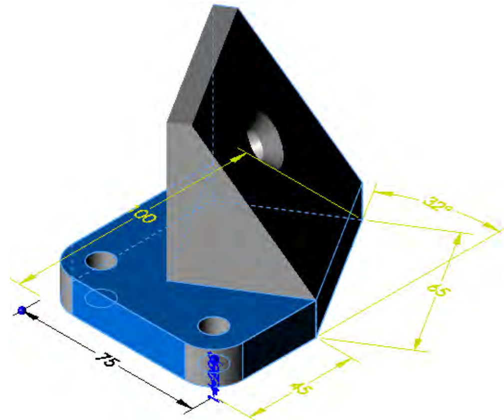


Exercise 43: SketchXpert

Use the SketchXpert to repair the part.

This lab reinforces the following skills:

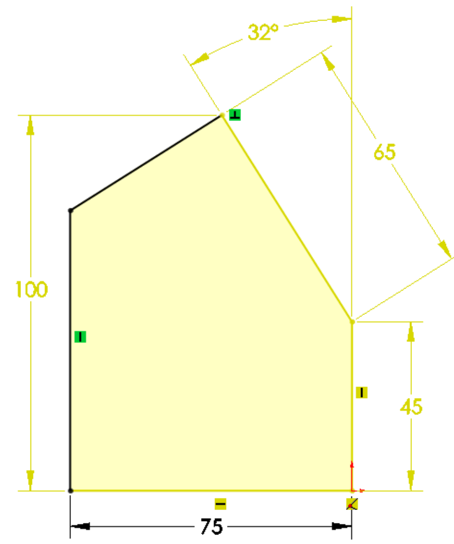
- *SketchXpert* on page 313.



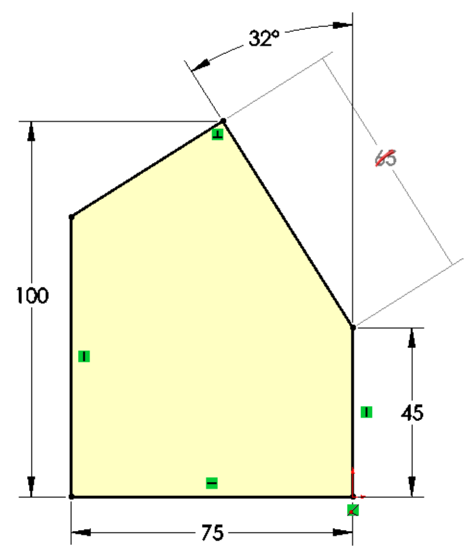
Procedure

Open an existing part and named `SketchXpert`. Repair the sketches as shown in the following steps.

1 Edit `Sketch1`.
Expand the `Base-Extrude` feature and edit the sketch `Sketch1`. Click **View Normal To** to orient the sketch as shown.

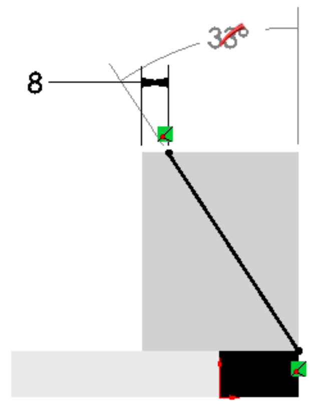


2 Solution.
Start the **SketchXpert** and click **Diagnose**. Choose the solution shown at right.

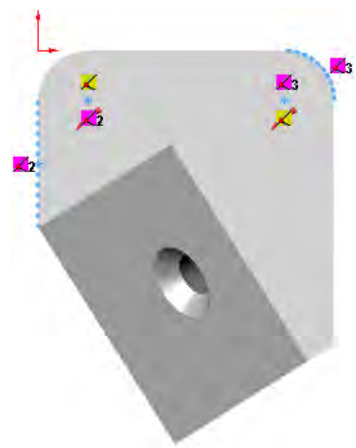


3 Other sketches.
Complete repairs to the remaining two sketches using SketchXpert.

Edit `Sketch3` from the `Cut-Extrude1` feature. Select the solution shown.



Edit `Sketch9` from the `Ø10.0 (10) Diameter Hole1` feature. Use **Manual Repair** and delete the relations shown.



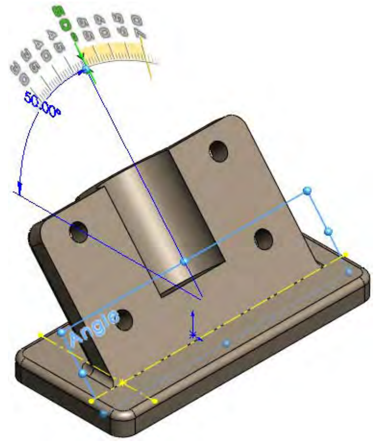
4 Save and close the part.

Exercise 44: Instant 3D

Use the Instant 3D tools to create, edit and complete the part.

This lab reinforces the following skills:

- *Editing with Instant 3D* on page 325.
- *Instant3D Handles* on page 325.
- *Dragging Face Geometry* on page 325.
- *One Click Changes* on page 326.



Procedure

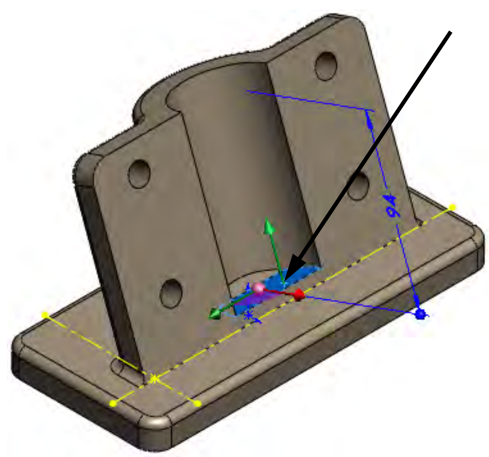
Open the existing part `Instant_Lab` and edit geometry as shown in the following steps.

Note

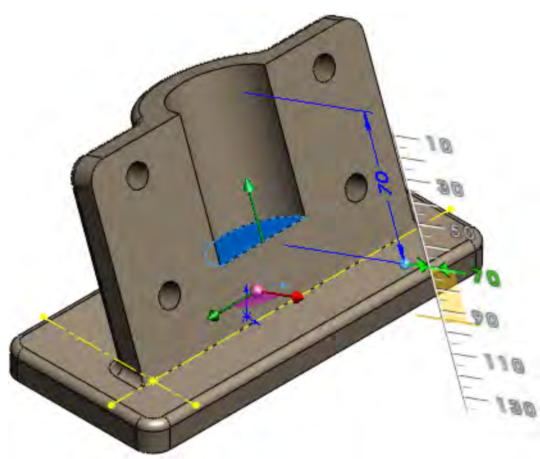
Repair any fillet errors that may occur.

1 **Open the part** `Instant_Lab`**.**

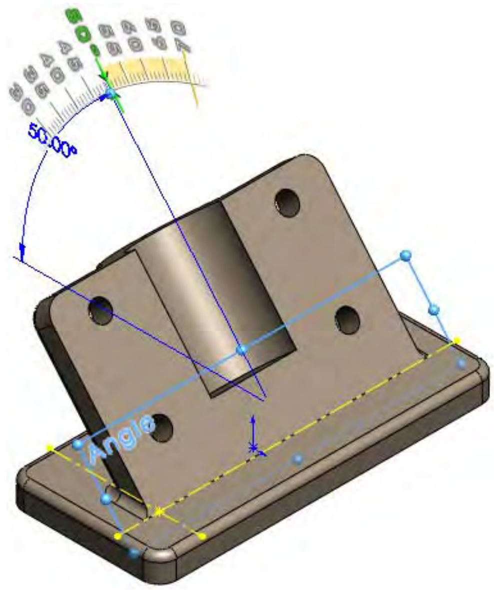
2 **Select face.**
Select the face of the `Stop` feature as shown.



3 **Drag handle.**
Drag the handle of the dimension to **70mm**.

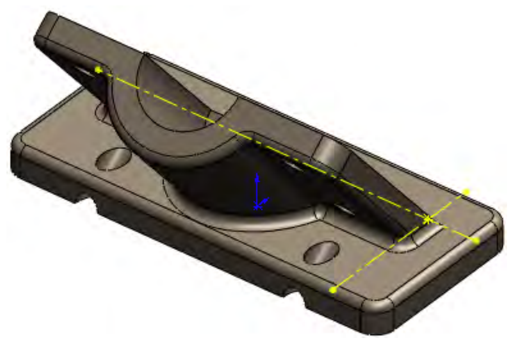


4 Drag angle.
Click the `Angle` plane and drag the handle to **50°** as shown.



5 Holes.
Change the display to show the rear of the model.

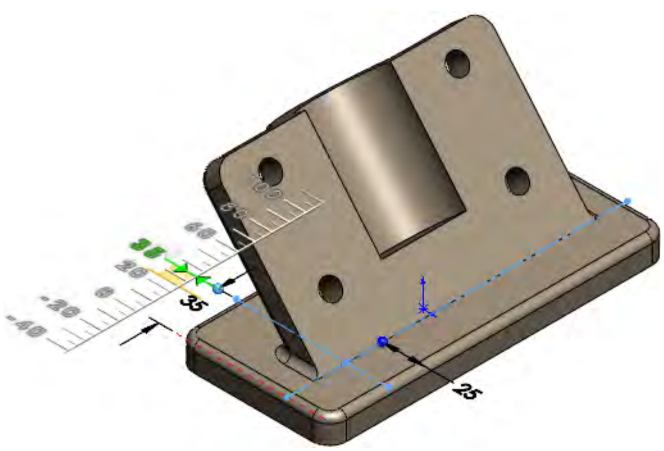
Edit the `Ø12.0 (12) Diameter Hole1` feature and change to an **Up To Next** end condition to prevent the extended cuts.



6 Driving sketch.
The `Layout` sketch was intended to drive the position of the `Thin Feature`.

Click the `Layout` sketch and drag the dimension to **35mm**.

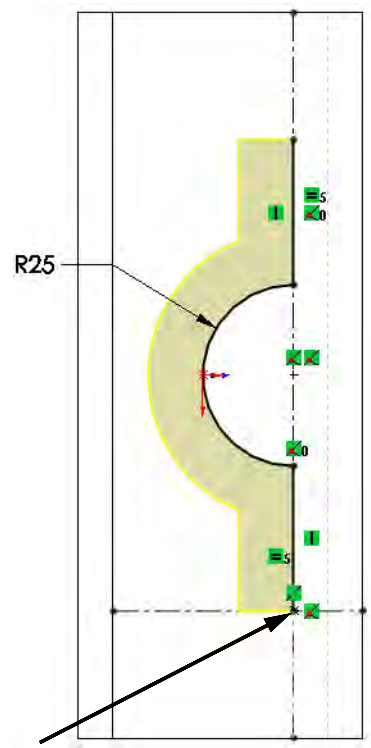
The change has no effect on the `Thin Feature`.



7 Edit sketch.

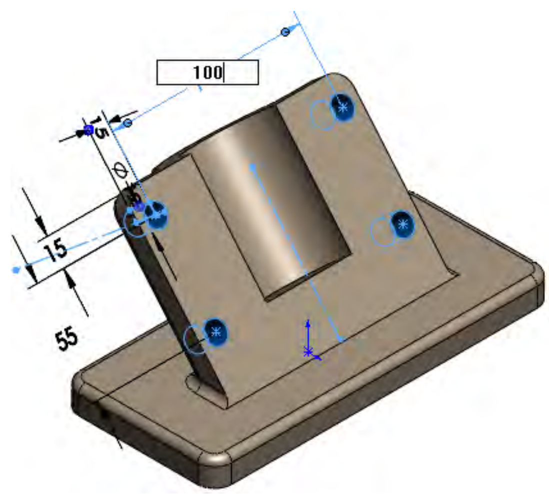
Edit the sketch of the `Thin Feature` and delete the **20mm** dimensions. Drag the endpoint to the intersection of the construction lines. Also. select both lines and add an **Equal** relation.

Exit the sketch.



8 Holes.

Click the `Ø12.0 (12) Diameter Hole1` feature and click the **110mm** dimension. Type **100**.



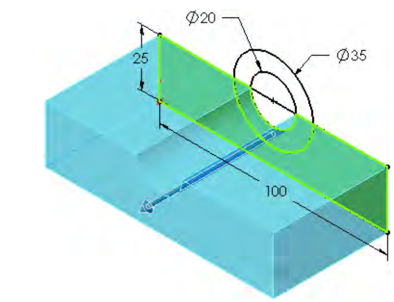
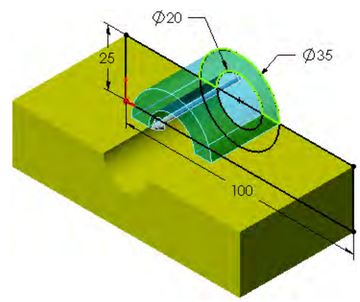
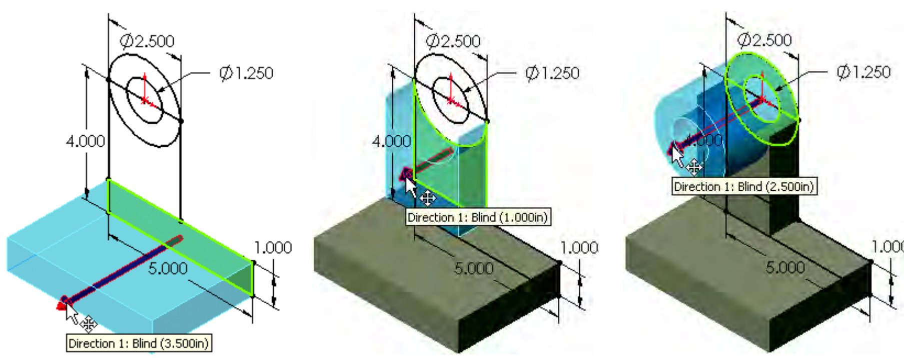
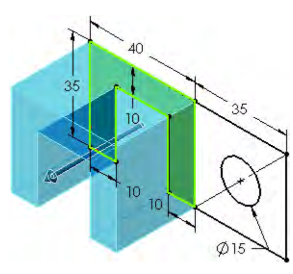
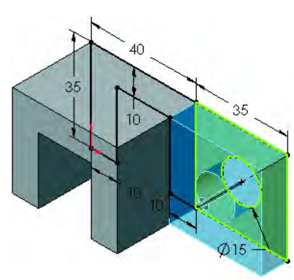
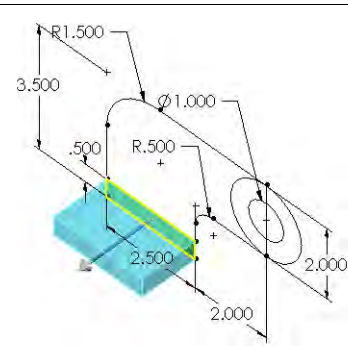
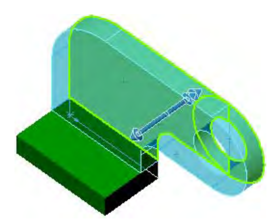
9 Save and close the model.

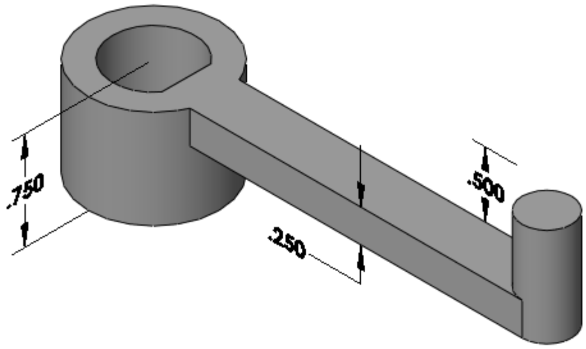
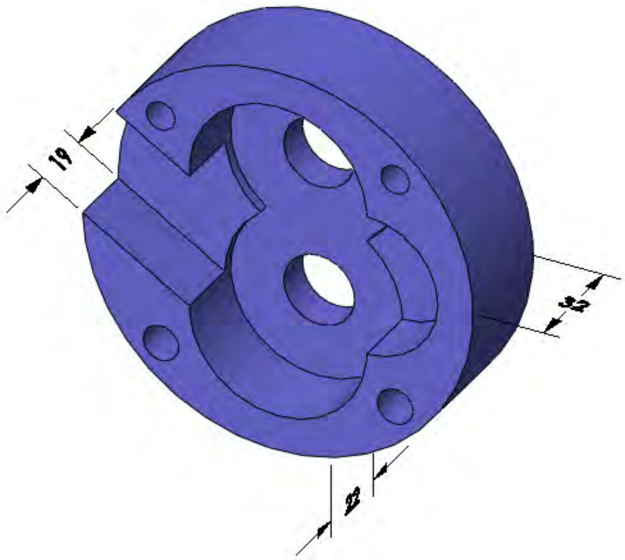
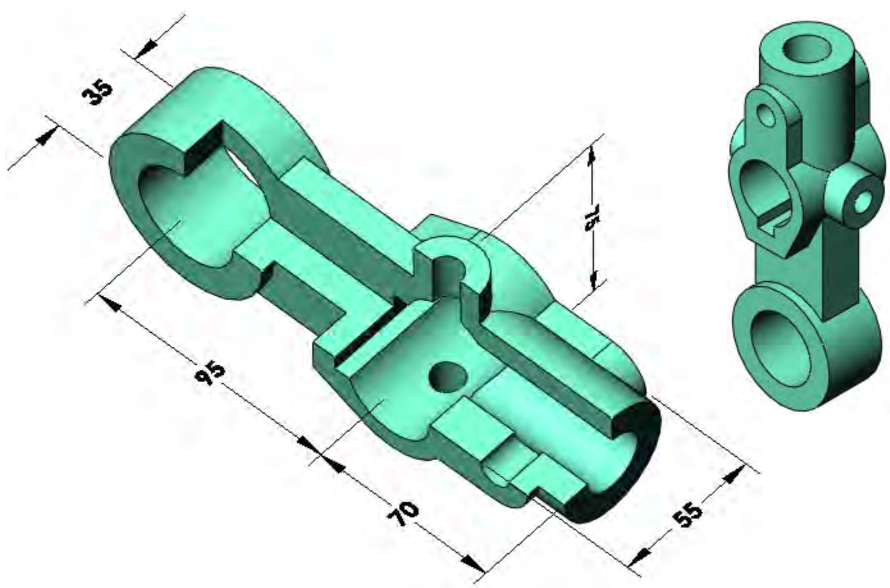
Exercise 45: Contour Sketches

Edit these parts using the information provided. Extrude profiles to create the geometry.

This lab reinforces the following skills:

- *Sketch Contours* on page 319.
- *Shared Sketches* on page 320.

#1 **Depth: 50mm and 30mm**		
#2 **Depth: 3.5”, 1” and 2.5”**		
#3 **Depth: 30mm and 10mm**		
#4 **Depth: 1.5” and 0.5”**		

Handle Arm	
Oil Pump	
Idler Arm	

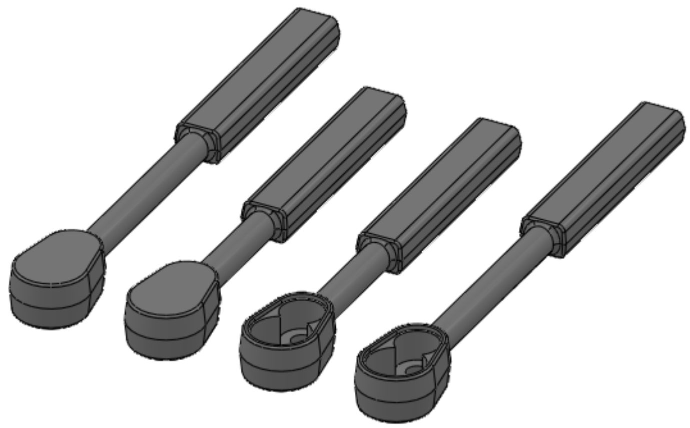
Lesson 10
Configurations

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.
- Use configure feature to create and edit configurations.
- Suppress and unsuppress features.
- Change dimension values by configuration.
- Suppress features by configuration.
- Link dimension values together to capture design intent.
- Create equations.
- Understand the ramifications of making changes to parts that have configurations.
- Use the design library to insert features into a part.

Configurations

Configurations allow you to represent more than one version of the part in the same file. For example, by adding 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.

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.

Methods to Create Configurations

SolidWorks allows you several methods to create configurations. Convenience often dictates the method used, but the methods can be mixed as needed. The chart below lists options.

Method	Description
Manually add names	Right-click on the top level component or the blank space in the ConfigurationManager and choose **Add Configuration**. This creates a new configuration name. When you add a new configuration, that configuration becomes active. See *Adding New Configurations* on page 344.
Copy and Paste	Select a configuration name in the ConfigurationManager 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.
Configure Feature	Right-click on a feature, material or dimension to access **Configure Feature Material** or **Dimension** tool in the **Modify Configurations** dialog. Use the *<Creates a new configuration.>* cell to add a new configuration. See *Creating Configurations* on page 344.
Design Table	A design table uses Microsoft Excel to create configuration names and track the changes. Because it uses Excel, the power of that product can be used in the creation of the configurations.

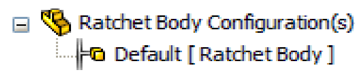
Procedure

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.**

Accessing the ConfigurationManager



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.

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**.

Creating Configurations

Configuring a feature or dimension means to alter it based on a configuration. For a feature, its suppression state (suppressed or unsuppressed) can vary by configuration. For a dimension, its value can vary by configuration.

Adding New Configurations

Configurations can be created manually. There are several options beyond the **Configuration name** that you can set in the new configurations.

Where to Find It

- Position the cursor within the ConfigurationManager and from the right-mouse menu, choose **Add Configuration**...

Bill of Materials Options

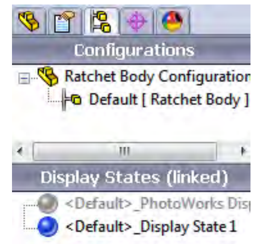
Sets the name that should appear under Part Number in a BOM.

Advanced Options

The advanced options include rules for creation of new features and color settings. Parent/Child options are for assemblies only.

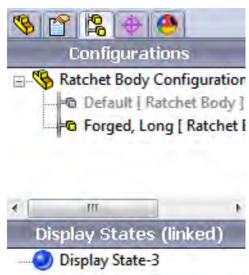
- **Suppress new features and mates**
  This option controls what happens to newly created features when other configurations are *active* and this configuration is *inactive*. If checked, new features added with other configurations active are suppressed in this one.
- **Use configuration specific color**
  Allows for different colors for each configuration using the color palette. Different materials may introduce different colors.

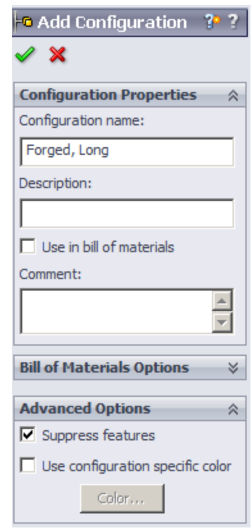
2 ConfigurationManager.
Click the ConfigurationManager tab.



3 Add new configuration.
Place the cursor in the **Configurations** area and right-click **Add Configuration**.

Type the name `Forged, Long` and click **Suppress features** under **Advanced Options.** Click **OK**.





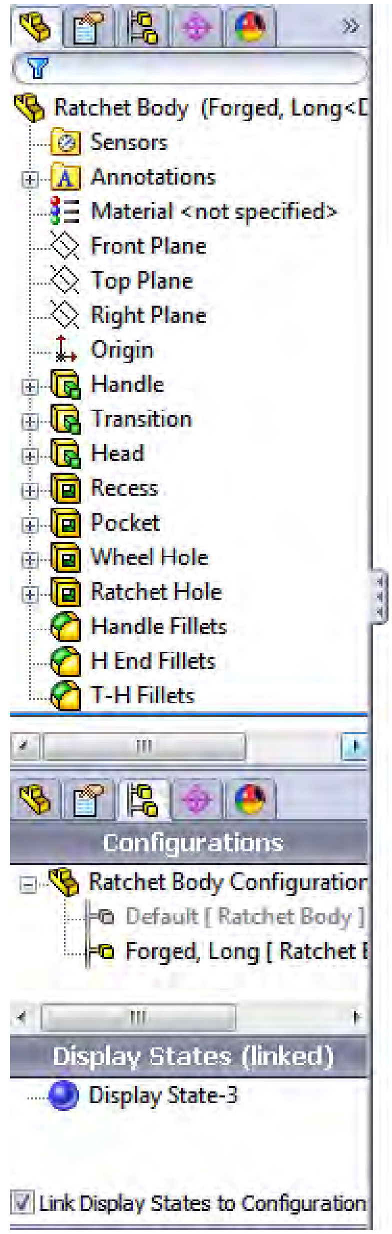
Tip

The active configuration is shown in color. The inactive configurations are greyed out.

Splitting the FeatureManager Window

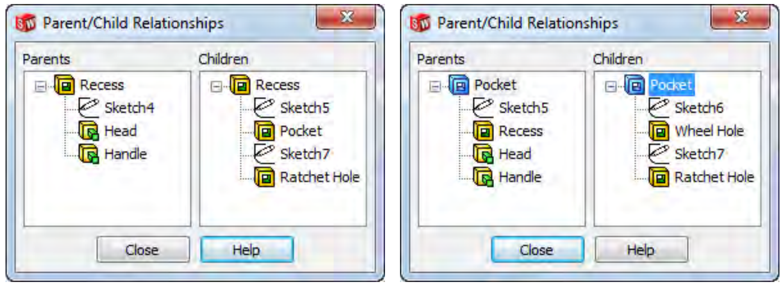
Many times it is efficient to be able to access *both* the FeatureManager design tree and the ConfigurationManager at the same time. This is particularly true when working with configurations. Rather than switch back and forth using the tabs, you can split the FeatureManager window top to bottom, creating two panes. One pane can show the FeatureManager design tree and the other can show the ConfigurationManager.

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.



4 Split.
Split the FeatureManager Window and arrange it like the image shown in *Splitting the FeatureManager Window* on page 346 with the FeatureManager design tree on the top and the ConfigurationManager on the bottom.

5 Check Parent/Child.
Right-click the Recess feature and select **Parent/Child**. From the dialog, right-click the Pocket feature and select **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.

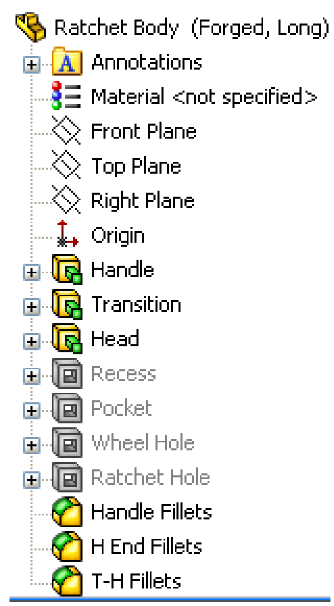
Unsuppress and **Unsuppress with Dependents** are used to reverse the effect of suppression on one (unsuppress) or more (unsuppress with dependents) features.

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 pull-down menu.
- Or, click **Suppressed** in the **Feature Properties** dialog box.

---

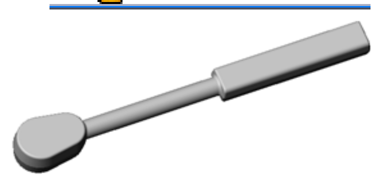
6 Suppress.
Close the open dialog. Right-click the Recess feature and select **Suppress**.



The system suppresses not only the Recess but also the Pocket, the Wheel Hole, and the Ratchet Hole. Why? Because the Pocket, and Ratchet Hole are all children of the Recess (see step **6** on page 347) and the Wheel Hole is a child of the Pocket.

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.

When the features are suppressed in the FeatureManager design tree, their corresponding geometry is suppressed in the model, too.



Copy and Paste Configurations

When you create a new configuration, the active configuration and settings are used to form the new configuration. You can also select any configuration, active or not, and copy it and paste it into the ConfigurationManager.

The name will be `Copy of <copied configuration>` and it will be inactive.

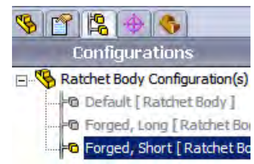
Changing the Active Configuration

To make a different configuration active, simply double-click on the one that you want in the ConfigurationManager. Only one configuration can be active at any time.

---

7 Copy and paste.
Select the `Forged, Long` configuration and click **Edit, Copy**. Click in the **Configurations** section and click **Edit, Paste**.

8 Rename.
Rename the configuration to `Forged, Short`.
Double-click it to make it the active configuration.



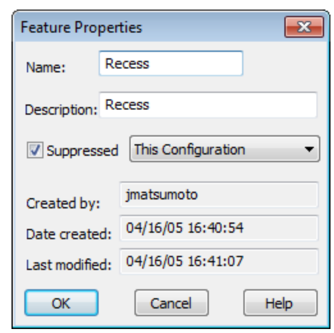
Other Ways to Configure

There are other ways to configure features and dimensions *after* the configuration names have been established.

Features

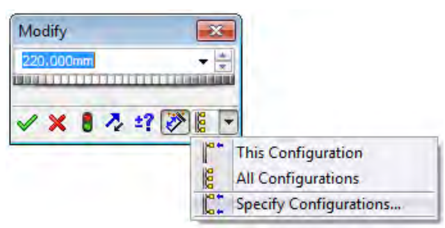
Features can be suppressed or unsuppressed in the active, specified, or all configurations using **Properties**.

Right-click the feature and select **Feature, Properties**. Check or clear **Suppressed** and select the configurations using the drop-down list.

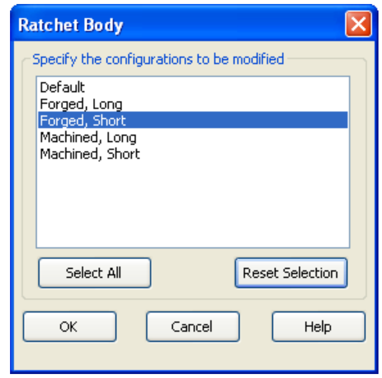


Dimensions

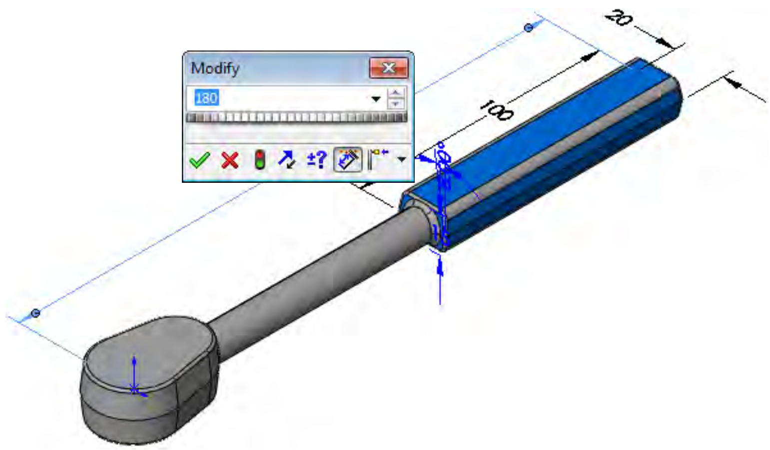
Double-click a dimension and change the value. In the drop-down, choose **This Configuration**, **All Configurations** or **Specify Configurations**.



If **Specify Configurations** is chosen, selected configurations can be chosen from the list of all configurations.



9 Change dimension value.
Double-click the `Handle` feature and double-click the **220mm** dimension. Type the value **180** and select **This Configuration**. Click **Regenerate** and **OK**.



10 Create new configuration.
Create a new configuration by copy and paste or **Add Configuration**. Name the configuration `Machined, Short`.

Unsuppress with Dependents

The options **Unsuppress with Dependents** and **Suppress with Dependents** can be used to unsuppress or suppress a feature and all dependents.

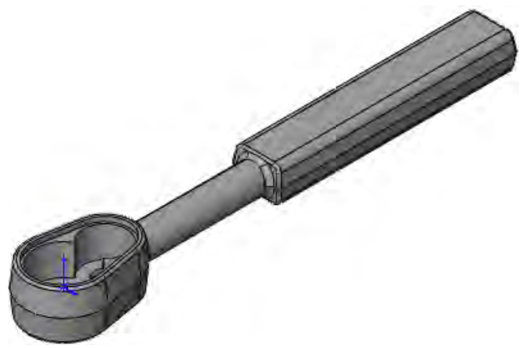
Note

Using unsuppress affects a single feature (see *Introducing: Suppress* on page 347) not the feature and children.

Where to Find It

- Click **Edit, Unsuppress with Dependents, This Configuration, All Configurations** or **Specified Configurations**.

11 Unsuppress.
Select Recess and click **Edit, Unsuppress with Dependents, This Configuration**.



Configuration Naming

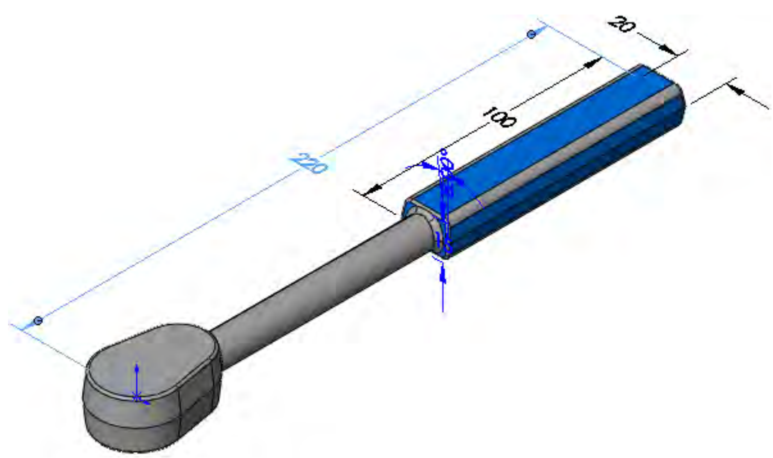
Configurations can be renamed in the same way as features. However, if a configuration is being referenced by another SolidWorks document, renaming that configuration can cause some difficulties.

A Better Approach

Instead of renaming the default configuration we will make a copy of it and then rename the copy.

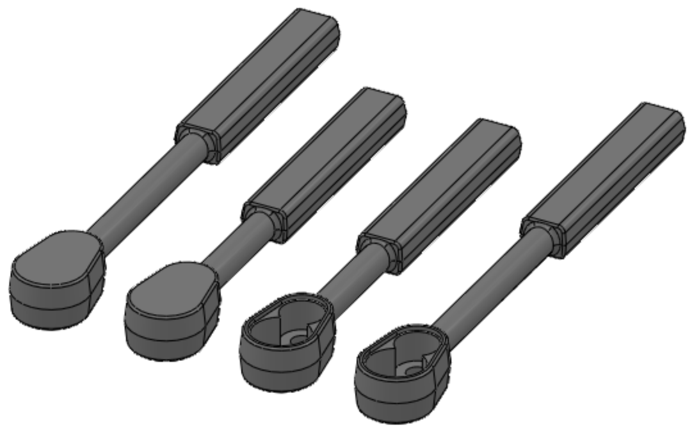
The configuration Machined, Long is the same as Default.

12 Copy and paste.
Copy the configuration Default and rename it to Machined, Long.



Completed Configurations

The completed configurations (not including Default) are shown below.

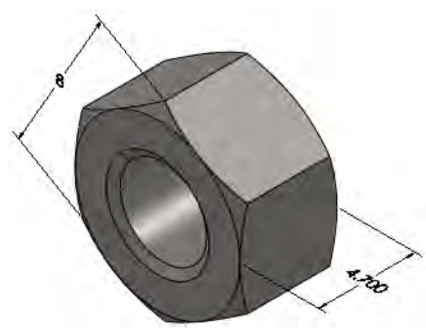


13 Save and close the file.

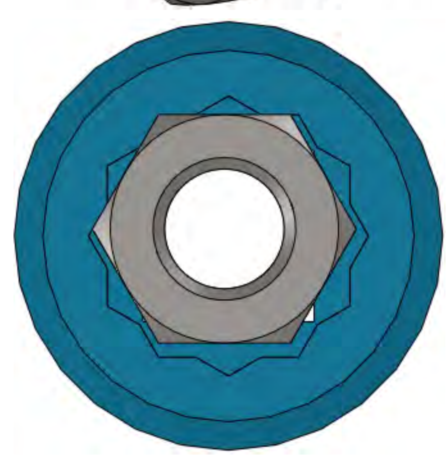
Using Link Values, Equations and Configure Feature

This section combines several useful tools in the process of creating a socket part. **Link Values** are used to set dimensions equal to one another. **Equations** are used to create a mathematical relationship between dimensions. **Configure Feature/Dimension** provides a table-driven method to create and manage configuration.

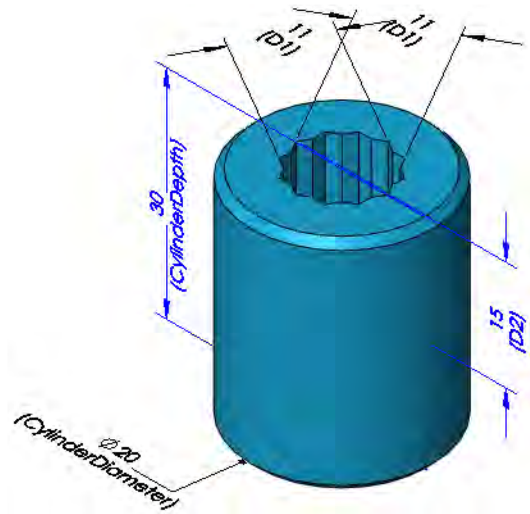
The design of this socket will be based upon an **ANSI Metric** size **M5 Hex Nut Style 1** component. The distance across the flat faces is **8mm** and the depth is **4.7mm**.



The cut in the socket will be larger than those dimensions to accommodate the nut.



1 **Open Socket.**
The Socket part (shown here with **Show Feature Dimensions** and **View, Dimensions Names** on) contains two cut features that represent the overlapping hexagon cuts. They retain their original names.



Note Right-click the Annotations folder to toggle **Show Feature Dimensions** on. Click **View, Dimensions Names**.

2 **Change value.**
Double-click the 6 Point feature and double-click the **11mm** dimension. Change the value to **8mm** and add **1mm** to the nominal value to represent the gap between a bolt head and the socket as shown.

We are using a unit value to make the configurations easy to work with. Click **OK**.

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.

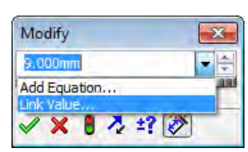
In this example there are two linear dimensions: one in each of the hexagon shaped cuts. Link values will be used to tie them together.

Where to Find It

- From the Dimension Modify dialog, choose **Link Value**.
- Right-click one or more dimensions and select **Link Values**.

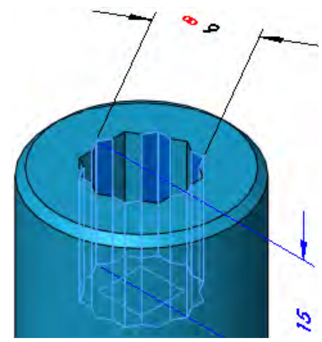
Note The dimensions being linked together must be of the same type. Link angular dimensions to other angular dimensions and so forth.

3 **Access link values.**
Double-click the same dimension as if to change the value. Using the drop-down menu, select **Link Value**.

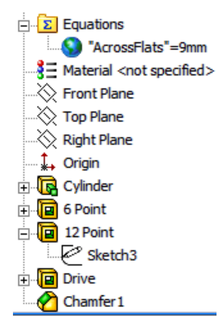


4 Name the link value.
In the **Shared Values** dialog, type the name `AcrossFlats` and click **OK**.

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.



6 Equation folder.
The link value is listed under the `Equations` folder in the FeatureManager design tree.



7 Add link value.
Double-click the sketch of the `12 Point` feature and double-click the **11mm** dimension. Using the drop-down menu, select **Link Value** and select `AcrossFlats` from the drop-down. The value of the existing link value is assigned to this dimension.

Rebuild the model.

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.

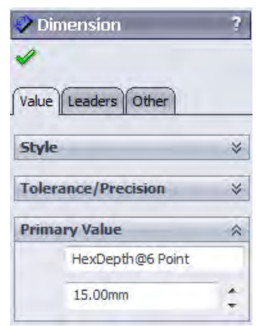
Note

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. Click a dimension, select the **Value** tab and click in the **Primary Value** to rename it. Only the portion before the @ sign is changed.



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, trigonometry 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 = 6

Introducing: Equations

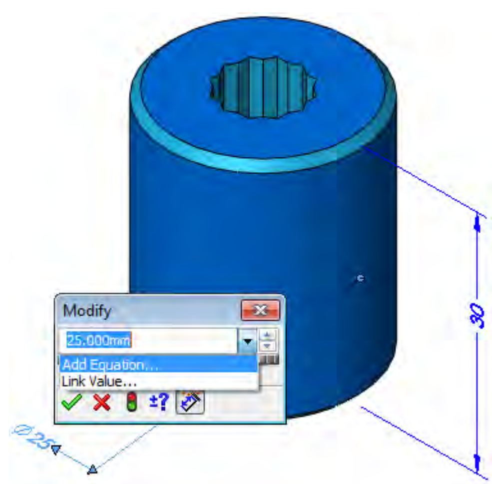
The **Equations** dialog can be used to add, edit, delete and configure equations.

Where to Find It

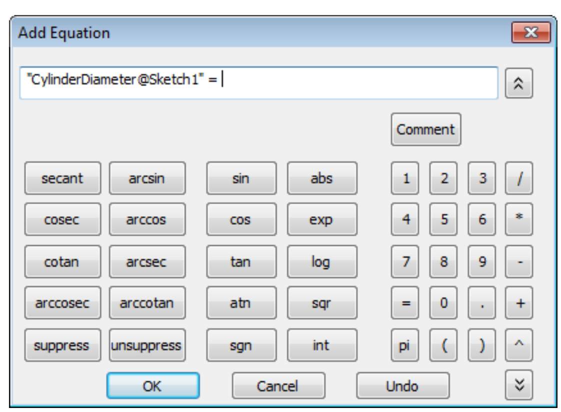
- Click **Equations** on the Tools toolbar.
- Or, from the **Tools** menu, click **Equations**.
- Or, right-click the Equations folder and choose an option.
- Or, from the Dimension Modify dialog, choose **Add Equation**.

---

8 Add Equation.
Double-click the Cylinder feature and the diameter dimension (**25mm**). In the dialog, choose **Add Equation** from the drop-down list.

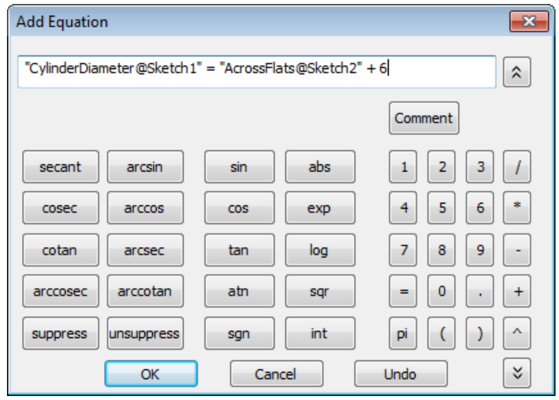


9 Dimension added.
The dimension is added to the new equation on the *left* side of the equals sign.

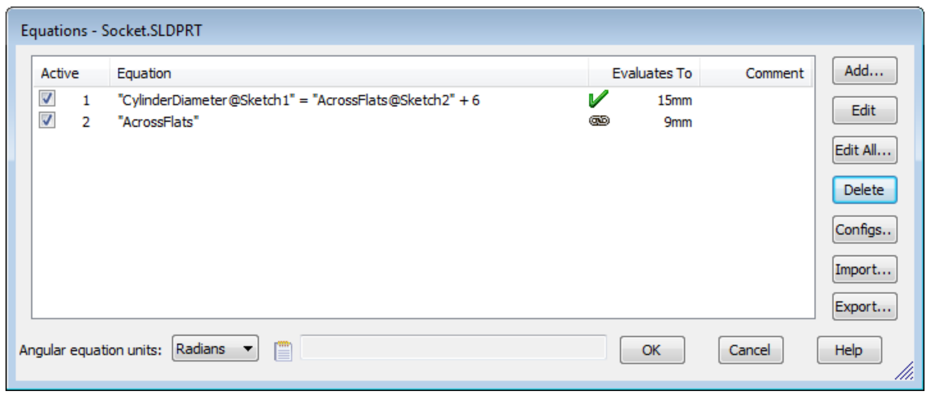


10 Complete equation.
Click either of the link value dimensions and add "**+ 6**" to complete the equation.

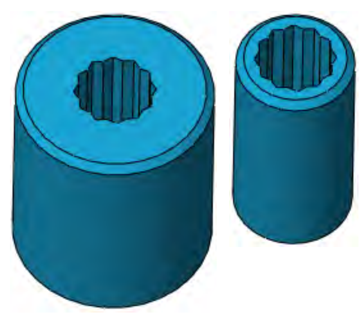
Click **OK** to add the equation.



11 List.
The current list of equations, including link values, are 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

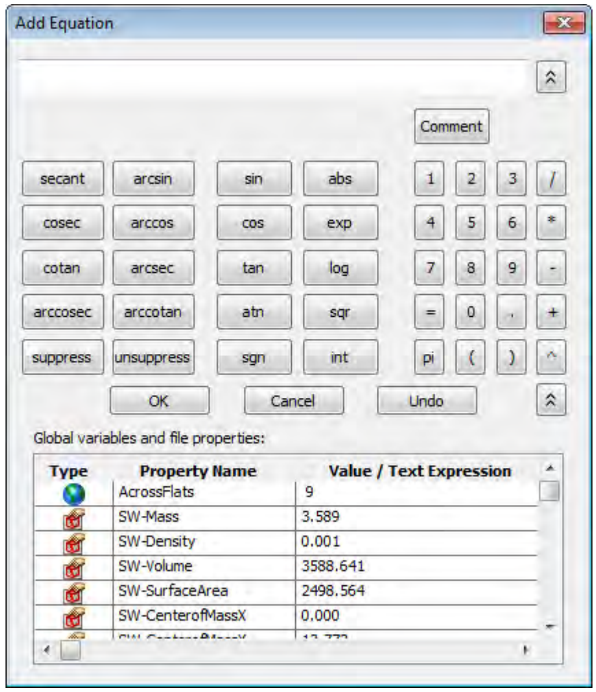
If your equations make use of angular dimensions, select **Radians** or **Degrees** as the **Angular Equation Units**.

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 and File Properties

Global Variables (or independent variables) and File Properties can be added and used within equations to represent yield strength, Poisson's ratio or other constants. They can be used within equations. A list of available global variables, default properties and custom properties can be accessed from the button.



Suppression Using an Equation

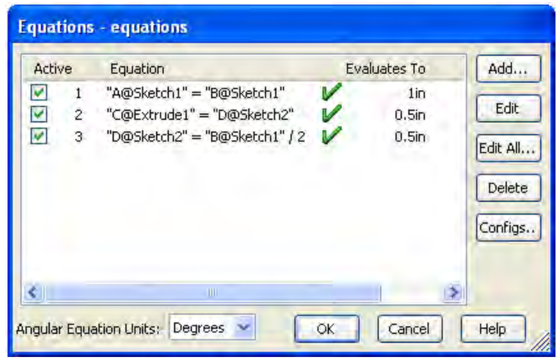
The **iif** statement can be used to suppress or unsuppress a feature based on a dimension value. For example, an equation to suppress the chamfer based on a diameter would be written like this:

```
"Chamfer1"= iif ("CylinderDiameter@Sketch1"<15,
"suppressed", "unsuppressed")
```

This means:

- Suppress the `Chamfer1` feature if the value of the `CylinderDiameter@Sketch1` dimension is less than or equal to **15mm**.
- Unsuppress the `Chamfer1` feature if the value of the `CylinderDiameter@Sketch1` dimension is greater than **15mm**.

A Few Final Words About Equations



Equations are solved in the order in which they are listed. If you try to add an equation and get this message:

"The variable D@Sketch2 is already used in the earlier equation and is getting modified. Please reorder OR change the equations to avoid multiple rebuilds." It means that your equations are in the wrong order. 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 would not be until the second rebuild that this new value for D would be used to compute a new value for C. Reordering the equations thus: A=B, D=B/2, and C=D solves the problem.

Configure Dimension/ Feature

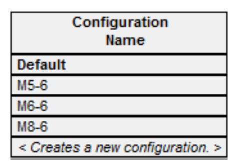
The **Configure Dimension/Feature** tool is another way to create configurations in the part. The interface uses a table to control configuration names, suppression states and dimension values.

Modify Configurations Columns

The dialog that appears when using either **Configure Dimension** or **Configure Feature** is **Modify Configurations**. It contains three types of columns: configuration, dimension or feature.

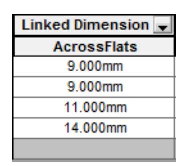
Configuration Column

The configuration column includes the current configurations and can be added by typing in the < Creates a new configuration. > cell.



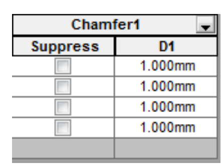
Dimension Columns

A dimension column is created when a dimension is configured or double-clicked into the table. Numeric values are typed into the cells.



Feature Columns

A feature column is created when a feature is configured or double-clicked into the table. The suppress checkbox can be checked or cleared. Related dimensions can be added using the pulldown menu on the upper right corner.



Configure Dimension

Configure a dimension using **Configure Dimension**. It can also be used to add new configurations.

Where to Find It

- From a dimension, right-click **Configure Dimension**.

Configure Feature

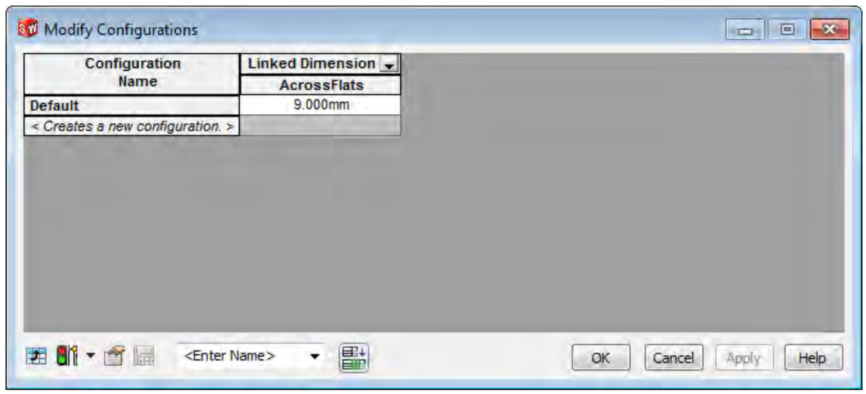
The **Configure Feature** tool is used to change features (suppression state) by configuration in a single dialog.

Where to Find It

- From a feature, right-click **Configure Feature**.

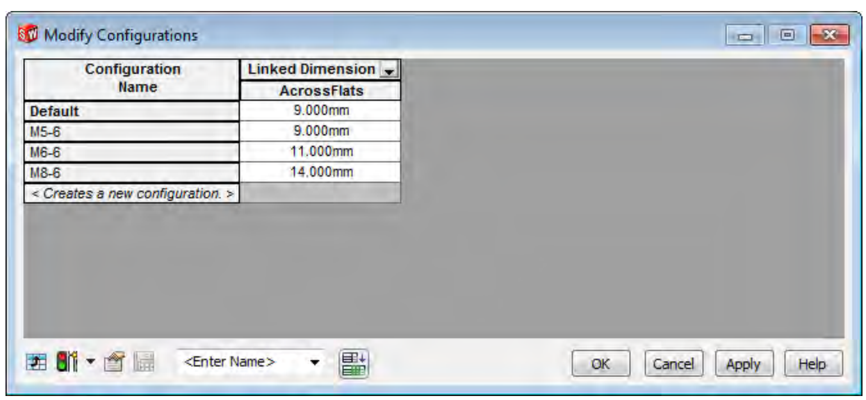
---

12 Modify dimension.
Double-click the `6 Point` feature. Right-click the **9mm** dimension and select **Configure Dimension**.



13 Create configurations and sizes.
Click in `< Creates a new configuration. >` and type `M5-6`. This refers to an M5 size that is a 6 point socket. Press **Enter** and type `M6-6` and then `M8-6`.

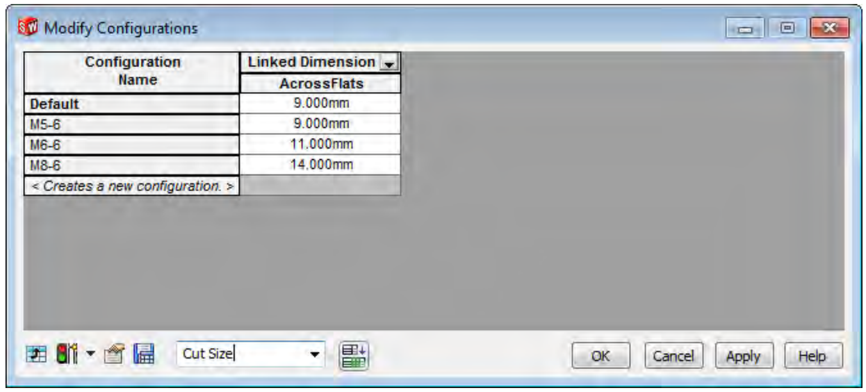
This dialog shows the value of the dimension by configuration. Click in the cells and change the values of the dimensions as shown.



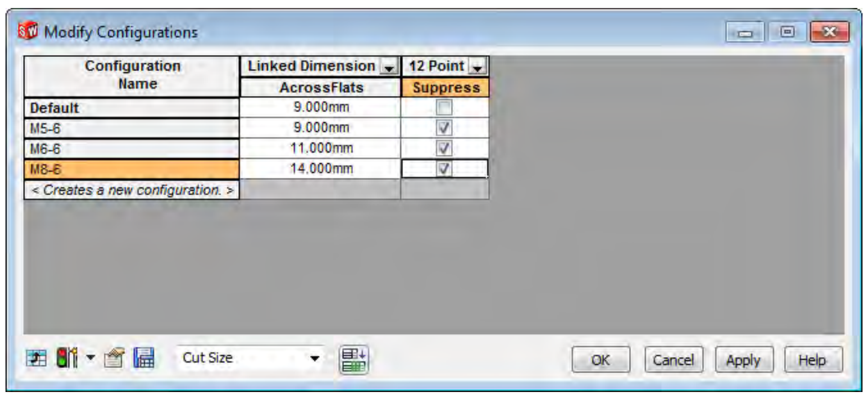
Tip

The dimension has the column title **Linked Dimension** because a link value was applied to it.

14 Name and save table.
Type in the name `Cut Size` and click **Save table view** and **Apply**.



15 Adding a feature as a column.
Double-click the feature `12 Point` in the FeatureManager design tree. Click **Suppress** for the `M5-6`, `M6-6` and `M8-6` configurations. Click **Save table view** and **OK**.

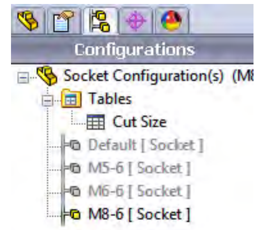


16 Resulting configurations.
Switch to the ConfigurationManager and double-click each configuration to make it active.

`M5-6` `M6-6` `M8-6`

17 Recall table.

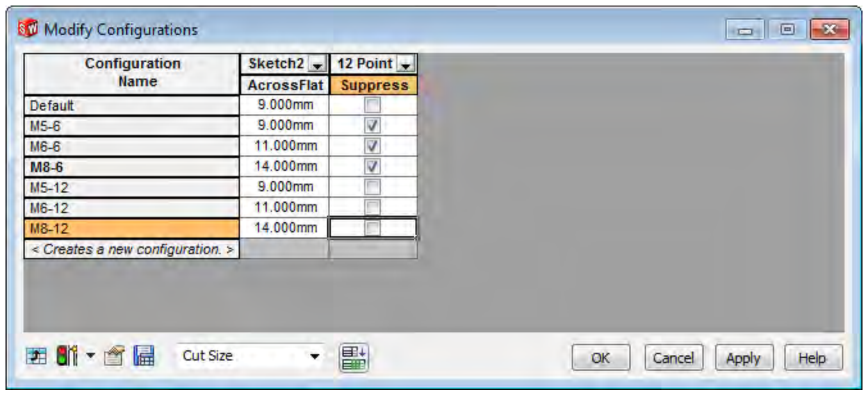
The table is stored in the ConfigurationManager under the `Tables` folder. Right-click **Show Table** to recall it.



18 Add more configurations.

Add the configurations `M5-12`, `M6-12` and `M8-12` as shown. Set the values and clear the **Suppress** column for them.

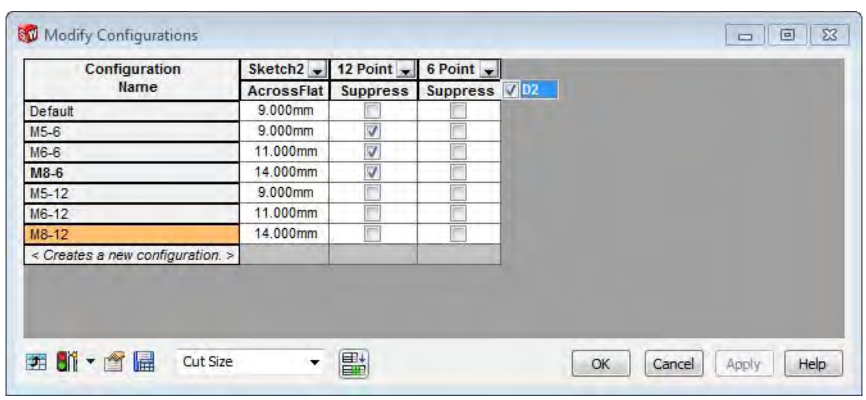
Click **Save table view** and **Apply**.



19 Dimension from a feature.

Double-click the feature `6 Point` in the FeatureManager design tree. Click the pulldown next to the feature name and select the dimension **D2**.

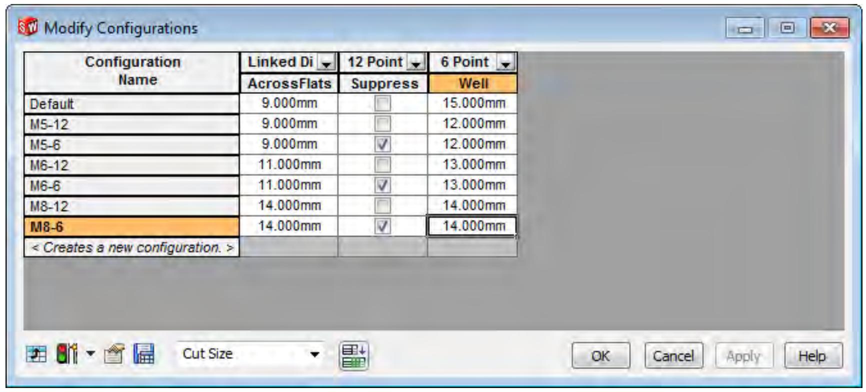
Click **Save table view**, **Apply** and **OK**.



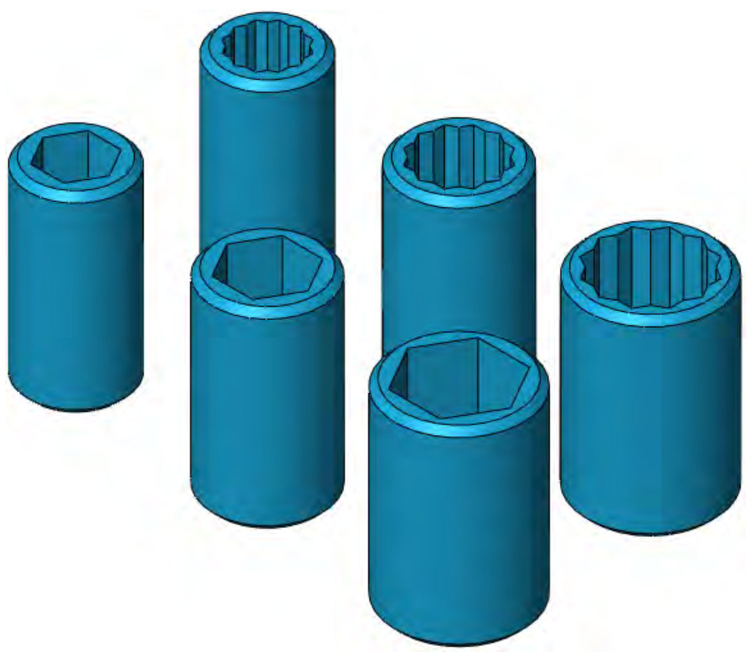
20 Delete and rename.
Double-click the table `Cut Size` to recall it.

Right-click in the title **Suppress** of the **6 Point** column and select **Delete**. Right-click in **D2** of the **6 Point** column and select **Rename** and type **Well**. Change the values as shown.

Click **Save table view** and **OK**.



21 Configurations created.
For each size there is a 6 and 12 point versions.



22 Save and close the part.

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.

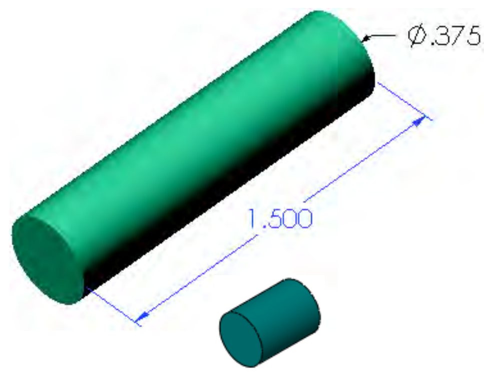
Performance Considerations

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 enables 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.

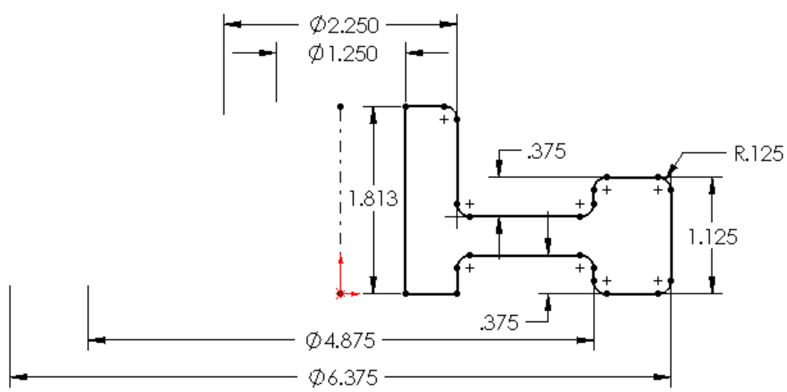
Similar parts that have the same basic shape can be defined as different configurations and used in the same assembly. The part shown at the right has two configurations. For an example showing how to use two different configurations of a part within an assembly see *Using Part Configurations in Assemblies* on page 425.



Modeling Strategies for Configurations

When you model a part that will be used with configurations, 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.

Editing Parts that Have Configurations

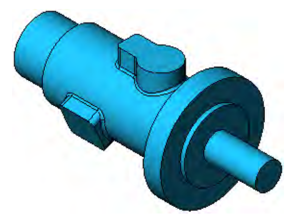
When configurations are added to a part, features may be automatically suppressed, dialogs list many additional options, and other strange things can happen. This section shows what happens when there are multiple configurations in the part being edited.

Note

End conditions, colors and materials can also be configured.

1 Open the part.

Open the part `WorkingConfigs`. This part has one configuration: `Default`.



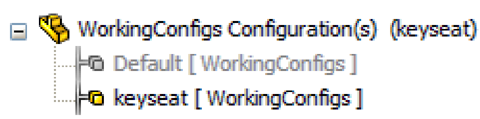
Configurations and new features will be added to the part.

2 Add configuration.

Use **Add Configuration** to add a configuration named `keyseat` and optionally, add a comment.

Click **OK**.

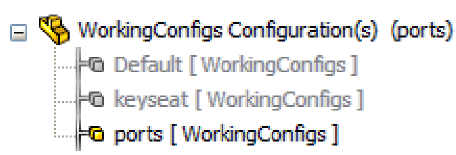
3 Added to list.



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.

4 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.
At this time, all three configurations are the same.

Tip

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.

Default Settings

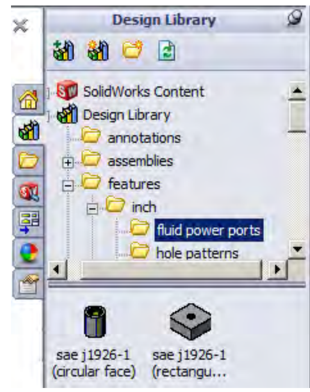
The first of three library features will be inserted using the default settings for location and size.

5 Folders.
Click the **Design Library** and the pushpin.

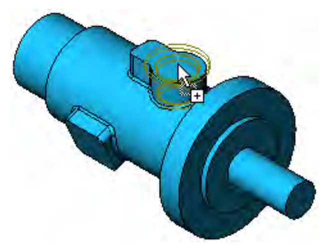
Expand the `features` folder.

Expand the `inch` folder.

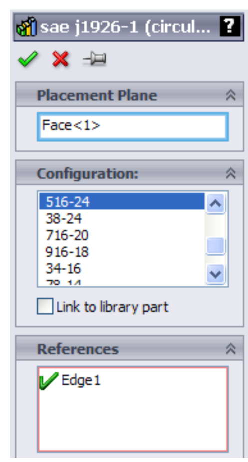
Click the `fluid power ports` folder.

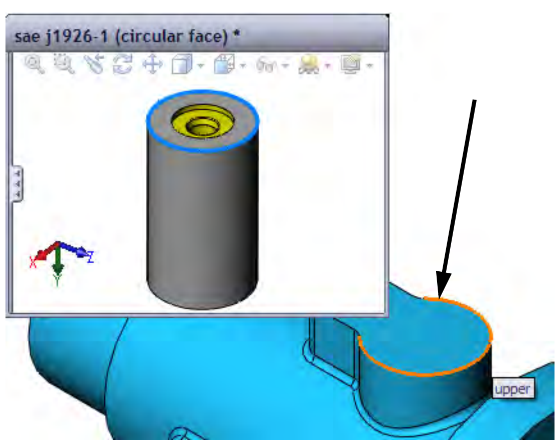


6 Drag and drop.
Drag and drop the `sae j1926-1 (circular face)` feature onto the planar model face as shown. The drop face is the **Placement Plane** for the feature.



7 Settings and selections.
Select the configuration `516-24` from the list. Select the `Edge1` (circular edge) reference as indicated in the preview window.





Click **OK**.

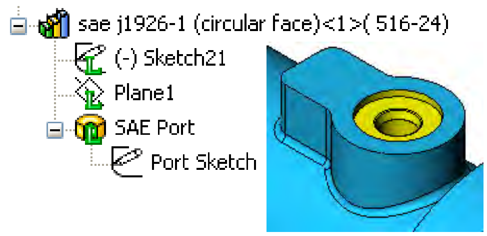
Note

The **Link to library part** option will create a link to update this part from changes in the library feature.

Tip

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 design tree as a library feature consisting of sketches, a plane and a cut.



Note

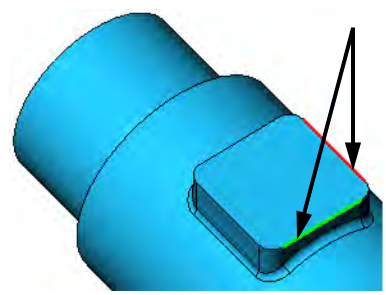
The “**L**” labels superimposed over the feature icons indicate a library feature.

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 278.

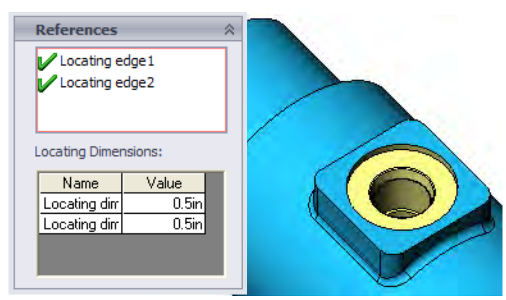
9 References.
Drag and drop the `sae j1926-1 (rectangular face)` feature onto the planar face. This feature requires the selection of two references, each being a linear model edge.



Select the `716-20` configuration. For the two **References**, select the edge shown in green followed by the edge shown in 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.

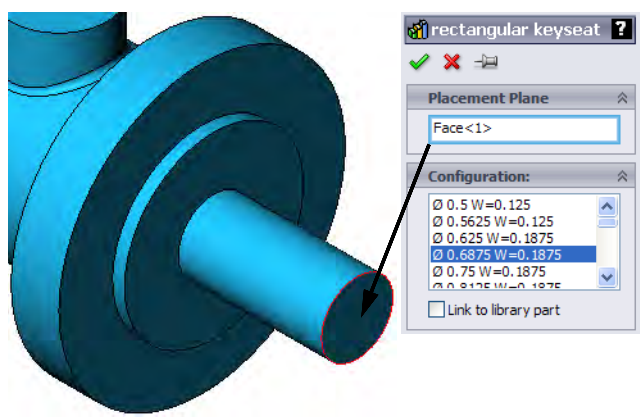
12 Active configuration.
Make the configuration keyseat the active one.

Dropping on Circular Faces

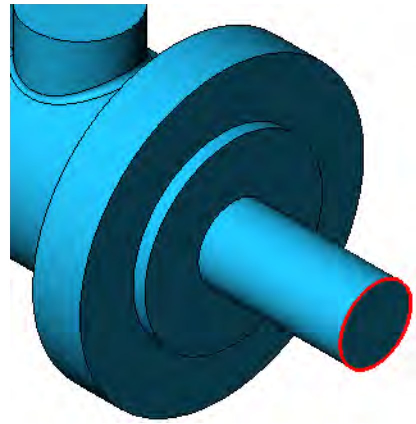
Some features attach to circular faces of the target model and require the first “drop” face to be that face. In these cases, the Placement Plane is selected after the drop.

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 the ∅ 0.6875 W=0.1875 configuration and the planar end face as the **Placement Plane**.

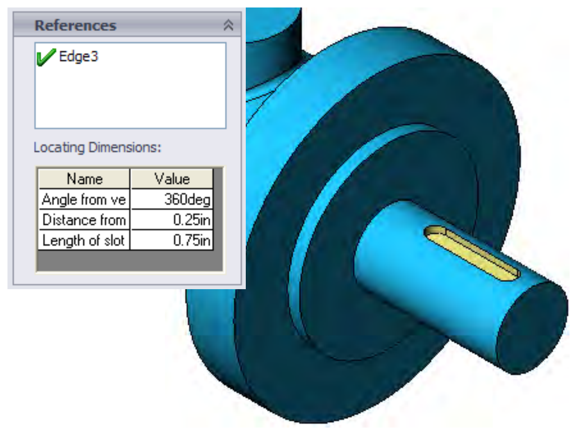


14 Reference.
Select the circular *edge* of the end face as the **Reference**.

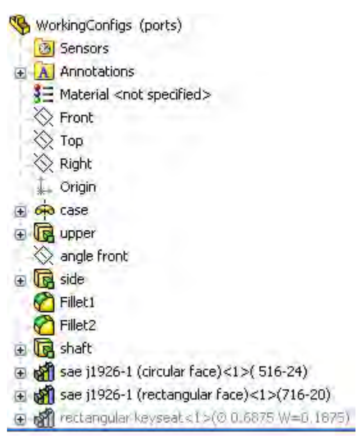


15 Locating dimensions.
Set the **Locating Dimensions** to the values shown.

Click **OK**.



16 Check configurations.
The new feature is *unsuppressed* in the active configuration (keyseat) but *suppressed* in all the others.



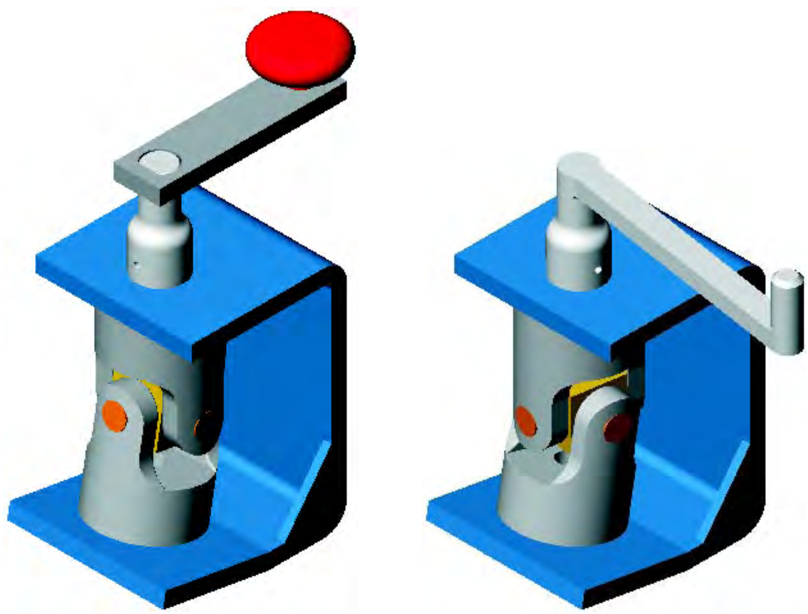
17 Save and close the file.

In the Advanced Course...

In the advanced course *Assembly Modeling*, the concept of **Configurations** is carried into assemblies.

Assemblies can have configurations that are created manually, through configure component or design tables. While part configurations focus on features, assembly configurations focus on components, mates, or assembly features. Assembly configurations can be used to control:

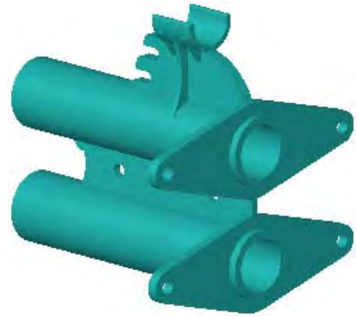
- Assembly Features
- Components
- Mates and Mate Dimensions



At the assembly level there are more options available to control one or more component instances.

Exercise 46: Configurations

Use an existing part as the basis for a series of configurations. Create different versions by suppressing various features in each configuration.

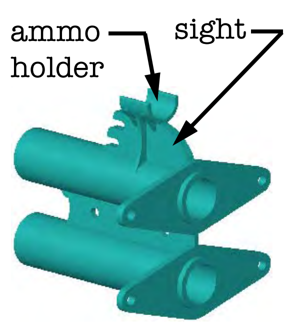
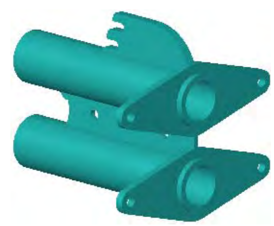
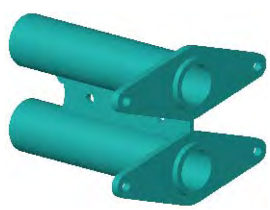
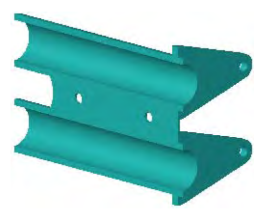


This lab reinforces the following skills:

- *Creating Configurations* on page 344.
- *Adding New Configurations* on page 344.
- *Introducing: Suppress* on page 347.

Configurations

Open the existing part config part. Create new configurations to match the conditions and names below. Add features to the model where required.

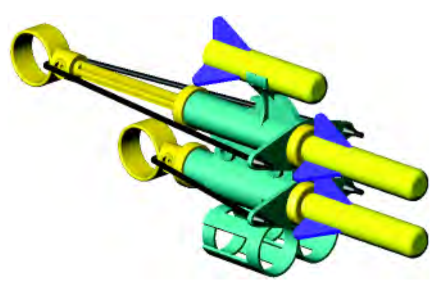
Best		Better	
Includes ammo holder and sight		Includes sight only	
Standard		Section	
Includes neither ammo holder nor sight		Shows a section cut through the Standard model	

Note

The Section configuration is created using a cut feature. To create the cut feature, activate the Standard configuration. Then use the Front plane and the command **Insert, Cut, With Surface...**, to cut the model.

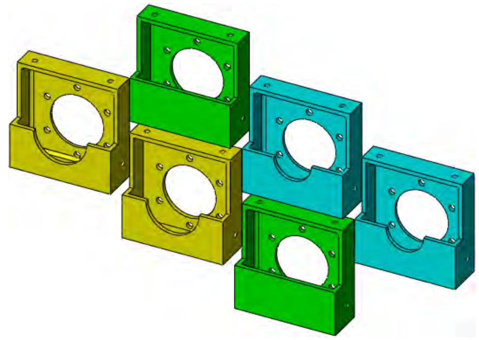
What is this Thing?

The part used in this example is the main twin barrel component from a toy that shoots soft, foam rockets.



Exercise 47: Working with Configurations

Using an existing part with configurations, add new features and modify others.



This lab reinforces the following skills:

- *Adding New Configurations* on page 344.
- *Introducing: Suppress* on page 347.
- *Editing Parts that Have Configurations* on page 365.

Procedure

Open the existing part `Working with Configurations`. The part contains seven (7) existing configurations. They are all copies of the `Default` configuration. Control the features as shown below.

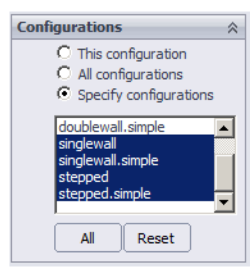
Suppression State

Set the suppression state of the following features.

Configuration	Fillets & Chamfers	Stepped sketch (cut)
doublewall	U	S
doublewall.simple	S	S
singlewall	U	S
singlewall.simple	S	S
stepped	U	U
stepped.simple	S	U

End Condition

To set the **End Condition** of the `center hole` feature use **Edit Feature**.



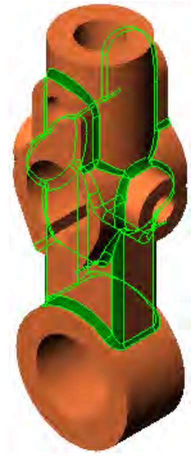
Configuration	center hole
doublewall	Through All
doublewall.simple	Through All
singlewall	Up To Next
singlewall.simple	Up To Next
stepped	Up To Next
stepped.simple	Up To Next

Exercise 48: Using Link Values

Create link values in an existing part and test it.

This lab reinforces the following skills:

- *Link Values* on page 365.



Procedure

Open the existing part named Link Values. Create a link value that makes all the fillets feature values equal.

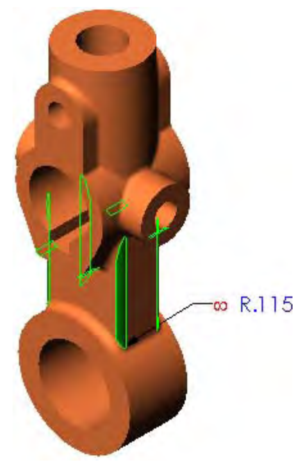
1 **Create link value.**
Create and apply a link value named All_fillets&rounds to the dimension of the Rounds feature.

2 **Apply link value.**
Apply the link value to the remaining three fillet features:

Fillets.1

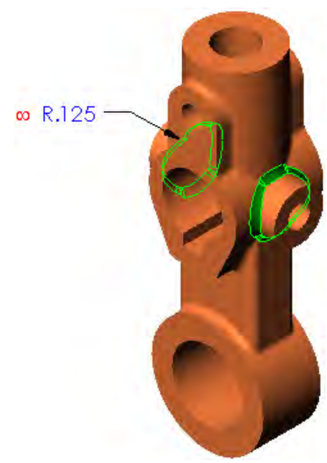
Fillets.2

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.**

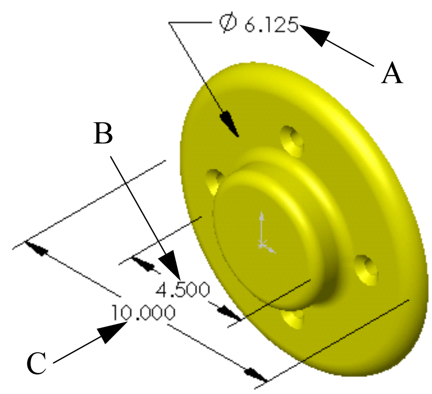


Exercise 49: Using Equations

Create an equation using an existing part and test it.

This lab reinforces the following skills:

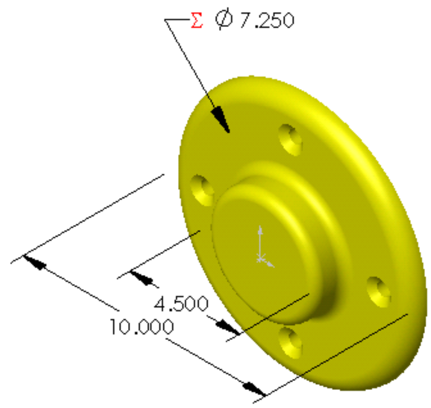
- *Equations* on page 367.



Procedure

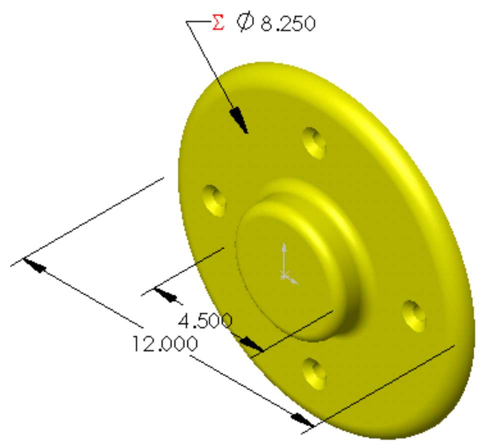
Open the existing part named `Using Equations`. The dimensions **A**, **B** and **C** shown above will be used to define the equation.

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.

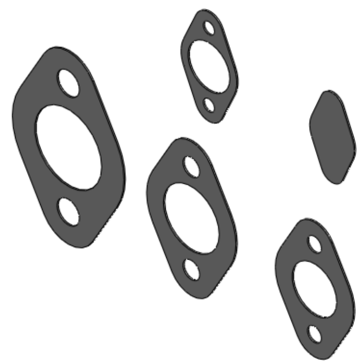


Tip

If you are having trouble, the equation format should be: `A=(C-B)/2+B`.

Exercise 50: Using Configure Dimension/ Feature 1

Use an existing part with configure dimension and configure feature to create new configurations.



This lab reinforces the following skills:

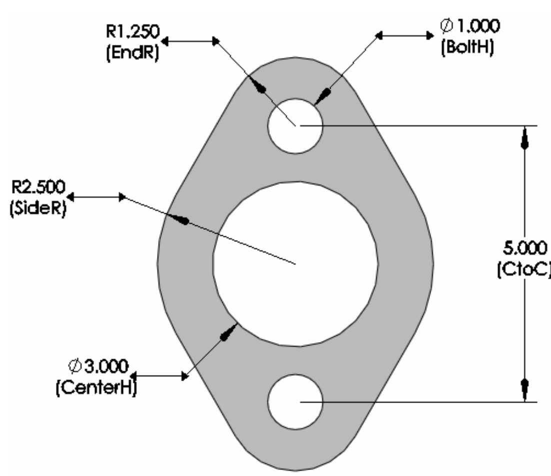
- *Configure Dimension* on page 358.
- *Configure Feature* on page 359.

Procedure

Open the existing part `Using Configure Feature`.

1 Configure Dimension and Feature.
Create configurations using configure dimension and configure feature. Create a table view with the settings shown.

Configuration Name	Sketch1			Holes	Sketch2	
	EndR	**SideR**	**CtoC**	**Suppress**	**BoltH**	**CenterH**
Size1	30mm	60mm	130mm	Clear	25mm	75mm
Size2	25mm	50mm	100mm	Clear	20mm	60mm
Size3	20mm	40mm	90mm	Clear	15mm	50mm
Size4	15mm	30mm	80mm	Clear	12mm	45mm
Size5	10mm	20mm	60mm	Click	10mm	30mm
Default	30mm	60mm	120mm	Clear	25mm	75mm



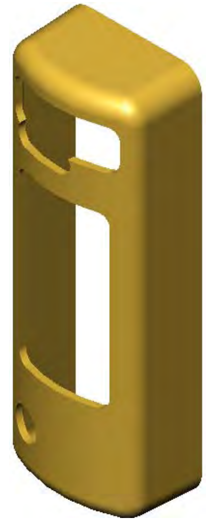
2 Save and close the part.

Exercise 51: Using Configure Dimension/ Feature 2

Use an existing part with configure dimension and configure feature to create new configurations.

This lab reinforces the following skills:

- *Configure Dimension* on page 358.
- *Configure Feature* on page 359.

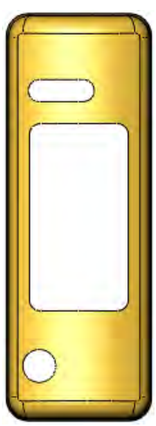
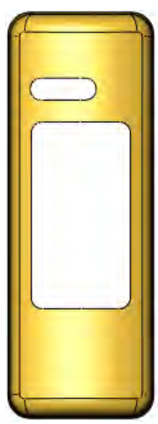
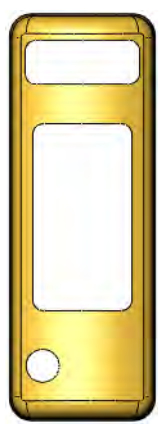
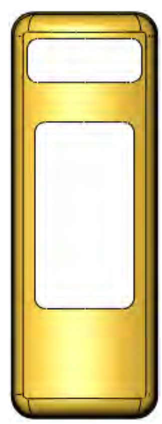
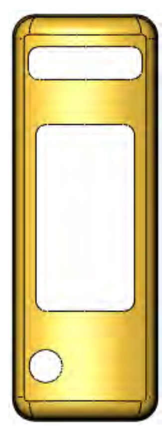
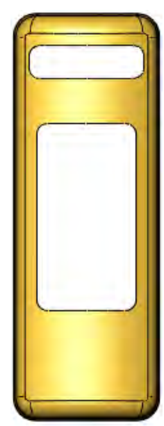


Procedure

Open the existing part Speaker.

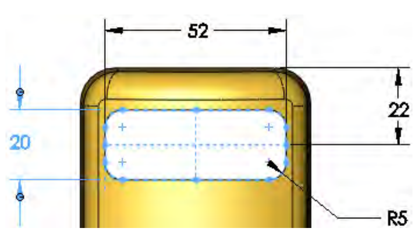
Configurations

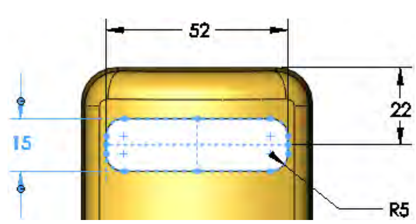
Suppress, unsuppress or modify dimensions of the features rounded tweeter, volume control and tweeter to create the following configurations.

100 Series		200 Series		300 Series	
100C	100S	200C	200S	300C	300S
					
C = Control, S = Slave					

Detail of tweeter

The tweeter feature size differs from the 200 to 300 series (right).





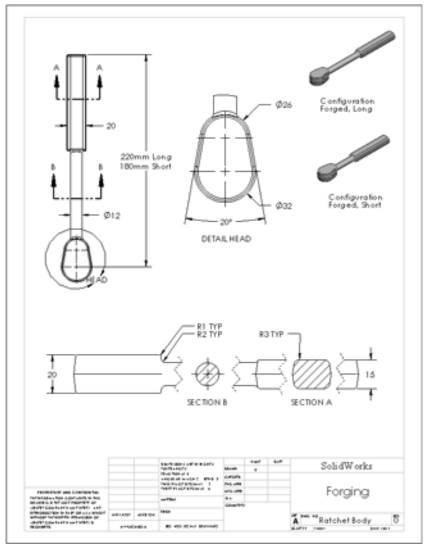
Lesson 11
Using Drawings

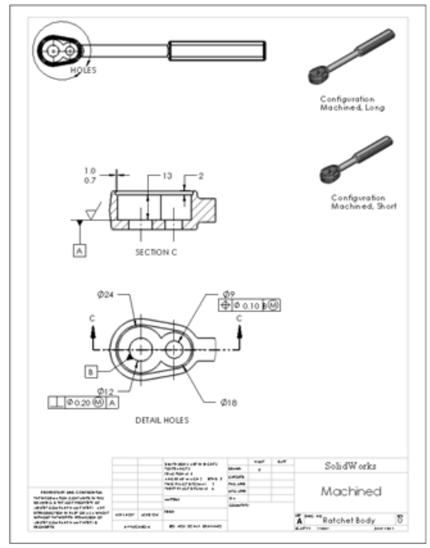
Upon successful completion of this lesson, you will be able to:

- Create several types of drawing views.
- Modify drawing views by alignment and tangent edges.
- Add annotations to a drawing.

More About Making Drawings

Drawings were first introduced in *Lesson 3: Basic Part Modeling*. In this section we will explore some additional detailing topics. These topics include: **Model Views**, **Section Views**, **Detail Views** and several **Annotation** types. In addition, multiple drawing sheets will be used to detail both the forged and machined configurations of the part.





Stages in the Process

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

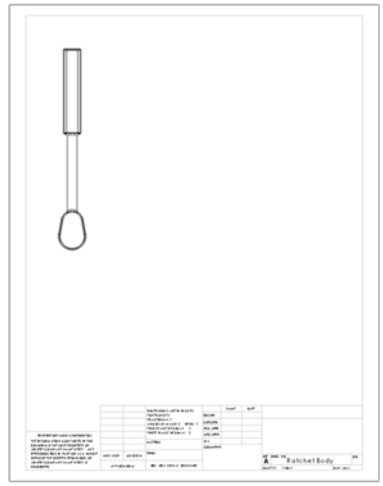
- **Views**
  Introductions to many commonly used drawing views including: section, broken, model, and projected views.
- **Annotations**
  Use annotations to add notes and symbols to the drawing.

1 Open the drawing Ratchet Body.

Open the drawing named Ratchet Body. It is an A (ANSI) Portrait sheet with one drawing view.

The other sheet settings are:

- **Type of projection** = Third angle
- **Scale** = 1:1



Section View

The **Section View** tool is used to create a new drawing view that is defined by cutting an existing view with a section line. The new view is automatically aligned to the parent view.

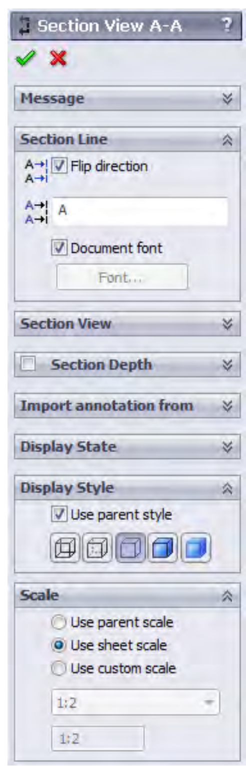
Where to Find It

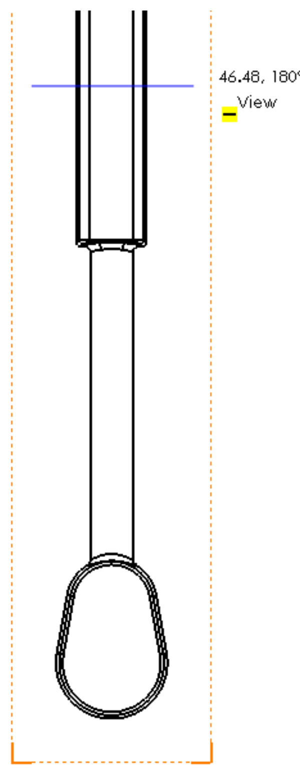
- Click **Section View** tool on the Drawing toolbar.

2 Sketch section view.

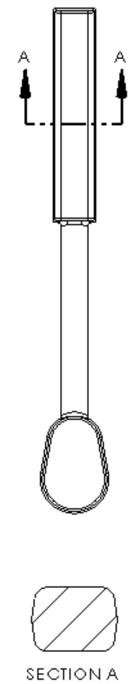
Click **Section View** and sketch the horizontal line as shown.

- **Flip direction** = selected
- **Import annotations** = cleared
- **Display style** = Click **Use parent style**
- **Scale** = Click **Use sheet scale**



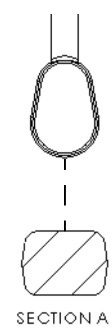


3 Place view.
Click below the view to place the section view.



View Alignment

Alignment is used to keep related drawing views aligned to one another by limiting movement. When you drag a view, dashed lines appear to show any existing alignment conditions. Alignment can be added or removed from any view.



Break Alignment

The automatic alignment between a source drawing view and a section, or projected view can be broken to move the view freely.

Where to Find It

- Right-click a view and select **Alignment, Break Alignment**.

4 Break alignment.
Right-click the section view and select **Alignment, Break Alignment**. The view is now free to move.

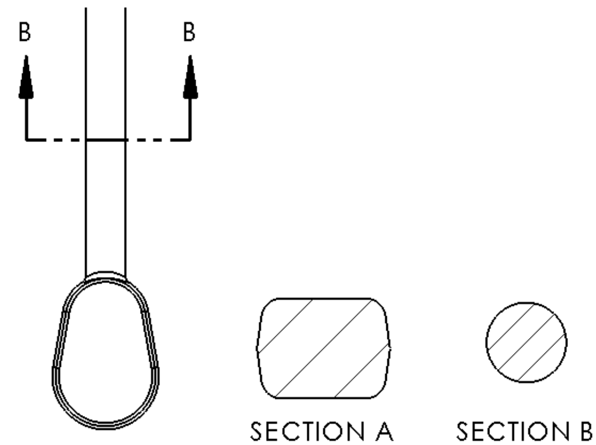
Tip

To move a view, click the edge or **Alt+**click anywhere in the view and drag.

5 Similar section.

Create another section view with similar settings to the previous one including the options **Partial section** and **Display only cut face(s)** selected.

After placing the section, break the alignment as in the previous step.



Model Views

The **Model View** creates a single view based on a predefined view orientation: Top, Front, Isometric and so on. The View Palette (*Drawing Views* on page 90) can also be used to create an orientation-based view.

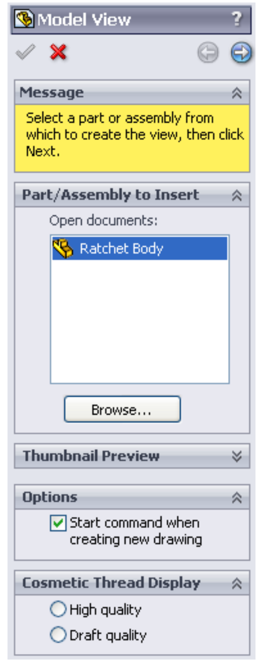
Where to Find It

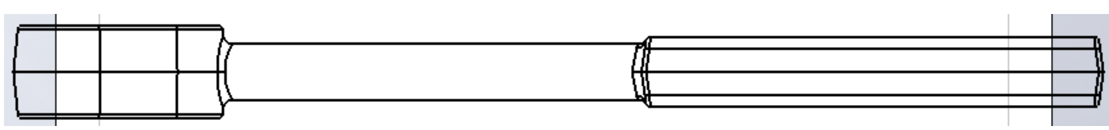
- Click **Model View** on the Drawing toolbar.

6 Model view.

Click **Model View** and select the Ratchet Body part. Click **Next**. Click *Right as the **Orientation** and **Hidden Lines Removed** as the **Display Style**. Click **Use sheet scale**.

Place the view just above the title block area and click **OK**. The view is wider than the sheet format borders.





Broken View

Broken Views, sometimes called interrupted views, make it possible to display a long part at a larger scale on a smaller size drawing sheet. This is done by creating a gap or break in the view using a pair of break lines.

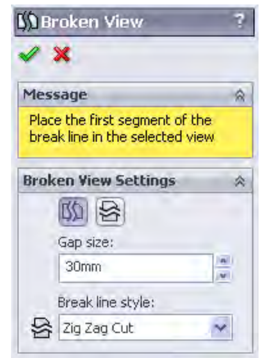
Where to Find It

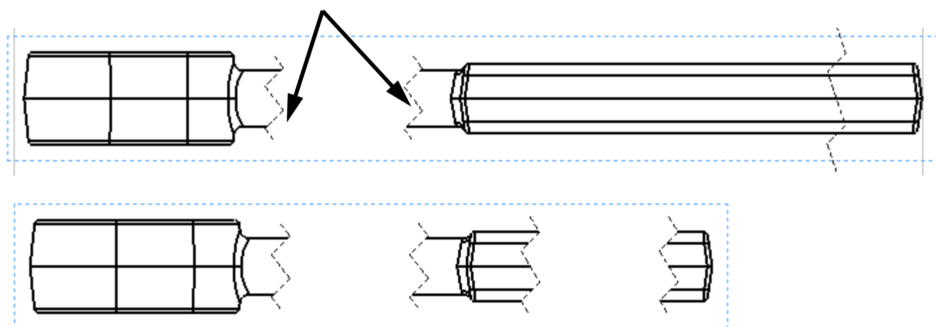
- Click **Break** on the Drawing toolbar.

7 Broken view.

Select the view and click **Broken View**. Click **Add vertical break line** and set the **Gap size** to **30mm**. Use the default **Zig-Zag Cut**.

Click in the view to create pairs or breaks. When the first pair is complete, the view breaks. Create a second break on the handle. Click **OK**.





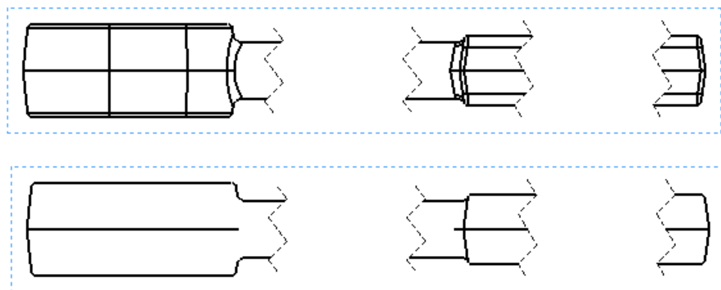
Tangent Edges

Tangent Edges are topological edges of faces that match in tangency. The most commonly seen tangent edges are the edges of fillets. They are often made visible in pictorial views but are removed from orthographic views.

Where to Find It

- Right-click the view and select **Tangent Edge** and an option.

8 Tangent edges removed.
Right-click in the view and select **Tangent Edge, Tangent Edges Removed**.



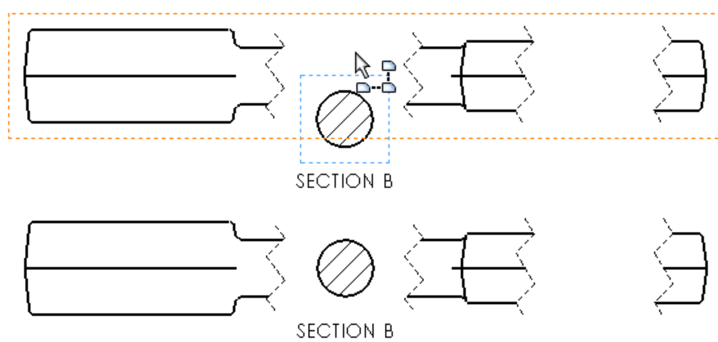
Aligning Views

Views can be aligned by origin or center. They can also be returned to the default alignment.

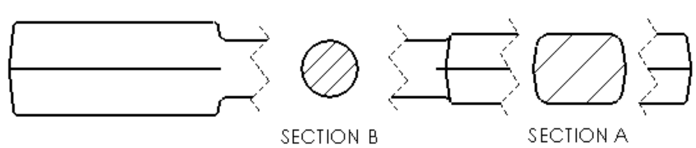
Where to Find It

- Right-click a view and select **Alignment**, and an option.

9 Align views.
Right-click in SECTION B and choose **Alignment, Align Horizontal by Origin**. Click in the broken view to complete the alignment.

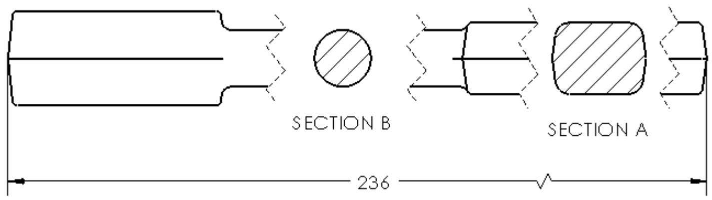


10 Complete revolved section.
Align SECTION A using the same procedure.



Note

If dimensions are added across the break lines, they are true and include the break symbol -√-.



Detail Views

Detail Views can be created using a closed sketched shape in an activated source view. The contents of the detail view are determined by what is enclosed within the sketch.

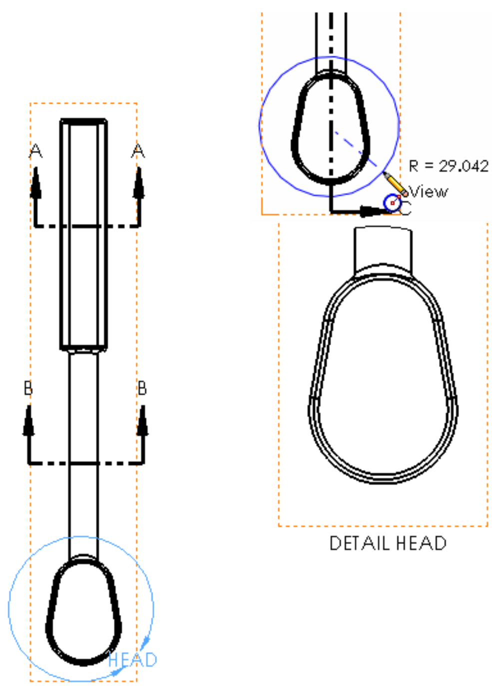
Where to Find It

- Click **Detail View** on the Drawing toolbar.

11 Detail view.

Click detail view and click **Draft quality** for **Cosmetic Thread Display**.

Sketch a circle in the view as shown. Name the view HEAD and click **Use sheet scale** and place the new view as shown.



Drawing Sheets and Sheet Formats

Drawing sheets, as mentioned earlier, represent the paper sheet. Contained within the drawing sheet is the sheet format that holds the title block and text.

Tip

The SolidWorks drawing file contains one or more drawing sheets which in turn can contain multiple views.

Drawing Sheets

The “paper sheets” are used to hold the views, dimensions and annotations and create the drawing.

Where to Find It

- Right-click on the drawing sheet and select **Edit Sheet Format** to access the sheet format.

Adding Drawing Sheets

In this lesson, multiple drawing sheets with multiple views will be created.

Tip

The new drawing sheet does not have to be the same size as the current drawing sheet.

Where to Find It

- Click the **Add Sheet** tab.

Sheet Formats

The border, title block and text used to add information to the drawing.

Where to Find It

- Right-click on the sheet format and select **Edit Sheet** to access the drawing sheet.

---

12 Add new sheet.

Click **Add Sheet** and add a new drawing sheet. Right-click the new sheet tab and choose **Rename**. Rename the sheet to `MACHINED`. Rename the original sheet to `FORGED`.

Projected Views

Projected Views are created by folding off an existing view in one of eight possible projections.

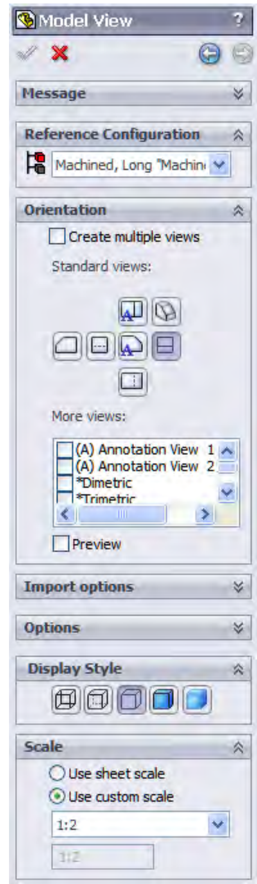
Where to Find It

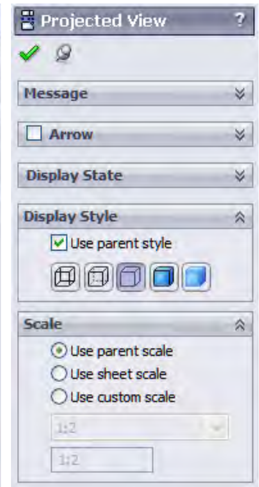
■ Click **Projected View** on the Drawing toolbar.

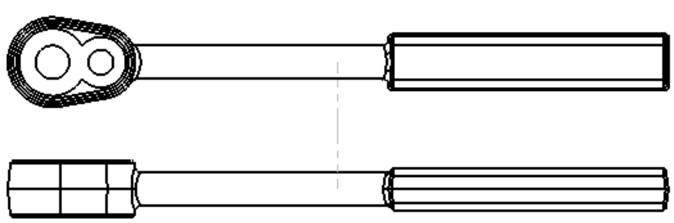
13 Projected view.

Create a **Model View** using the `Ratchet Body`. Under **Reference Configuration**, select `Machined, Long "Machined"` from the list.

Click `*Right` orientation and a custom scale of **1:2**. Move the cursor above the model view and click to place a **Projected View.** The projected view is the desired view.







14 Delete view.

Click the original model view (`*Right`) and press the **Delete** key.

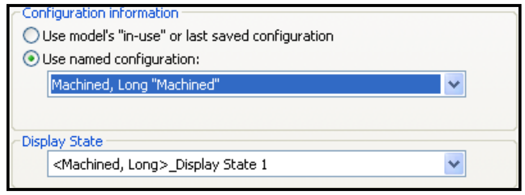
Drawing View Properties

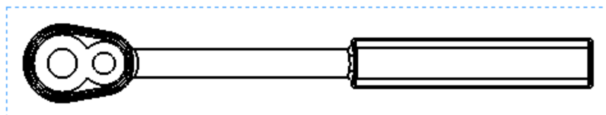
The **Drawing View Properties** dialog provides information about the view and allows you to change settings like the configuration of the component used in the view.

Where to Find It

- Right-click the view and select **Properties**.

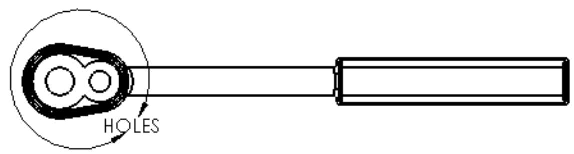
15 View properties.
Right-click the view and choose **Properties**. Check to see that under **Use named configuration**, Machined, Long "Machined" is used and click **OK**.

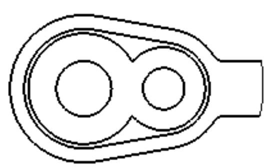




16 Detail view.
Add a detail view renaming it HOLES as shown.

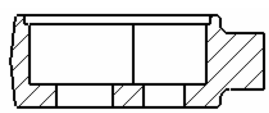
Remove the tangent edges.



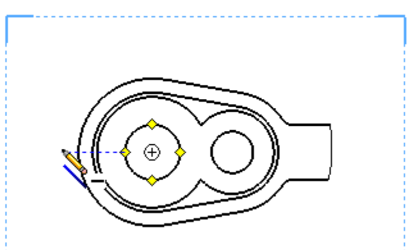


DETAIL HOLES

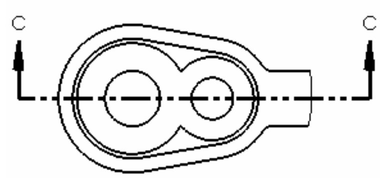
17 Section of detail.
Create a section through the centerpoint of an arc by "waking up" the centerpoint and sketching a line through it.



SECTION E



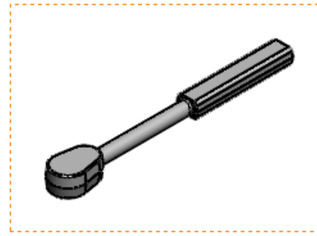
DETAIL HOLES



DETAIL HOLES

Rename section to SECTION E.

18 Model view.
Add a model view using the Forged, Long "Forged" configuration, *Isometric orientation, scale **1:4** and **Shaded with Edges** and place it in the upper right corner.



Annotations

Annotations are symbols used to enhance the drawing by providing additional information for manufacturing and assembly. Many types of annotations are available in SolidWorks. The most general type, the Note, is representative of the characteristics that annotations possess.

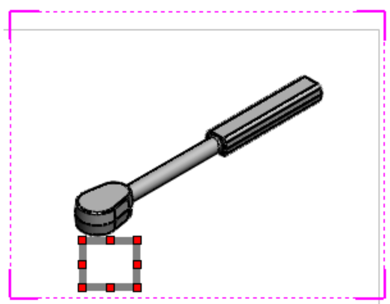
Notes

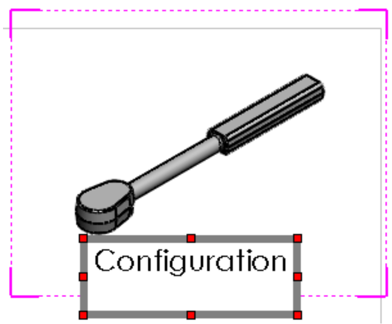
Notes are used to add text and labels to drawings.

Where to Find It

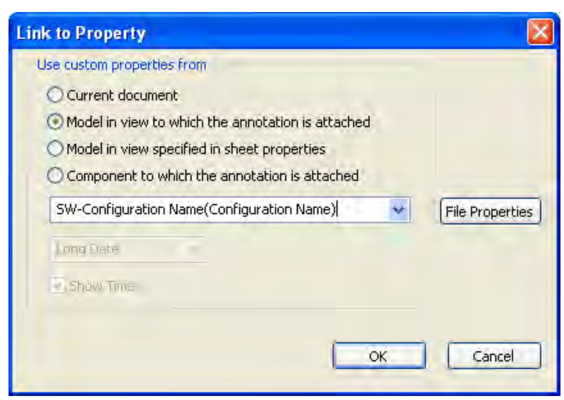
- Click **Note** from the Annotation toolbar.

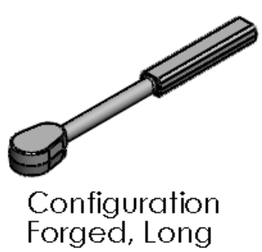
1 Note in view.
Double-click the model view. Click **Note** and click in the model view. Type Configuration and press the **Enter** key.





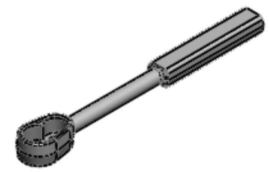
2 Link to property.
Click **Link to Property** and click **Model in view to which the annotation is attached**. Select SW-Configuration Name from the list and click **OK**.

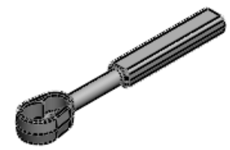




3 Copy and paste.
Click in the view and click **Edit, Copy**.
Click on the drawing and click **Edit, Paste**.

Using the drawing view properties, change the configurations to those shown here.





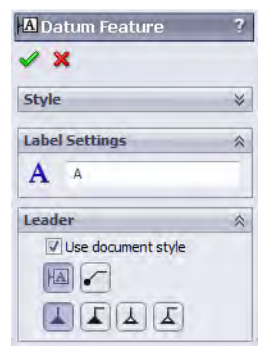
Datum Feature Symbols

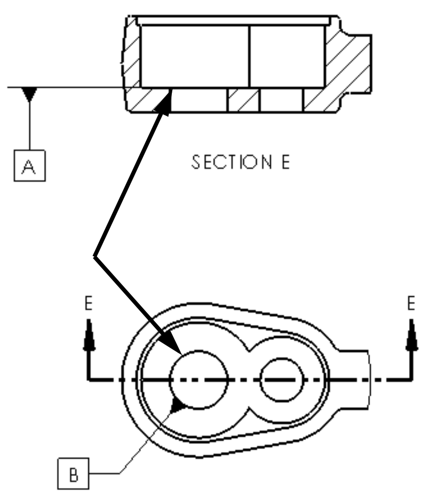
Datum Feature Symbols can be added to a drawing view on a surface that appears as an edge (including silhouettes) to identify datum planes in the part.

Where to Find It

- Click **Datum Feature Symbol** on the Annotation toolbar.

4 Add datums.
Click **Datum Feature** and click the line in SECTION E as shown. Move to the left and below to click and place the datum. Repeat the procedure for the arc in DETAIL HOLES.





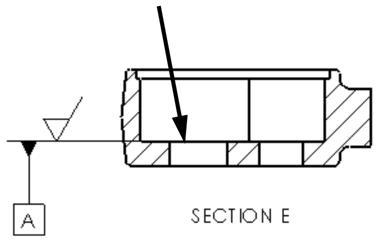
Surface Finish Symbols

You can specify the surface texture of a part face by using a **Surface Finish Symbol**.

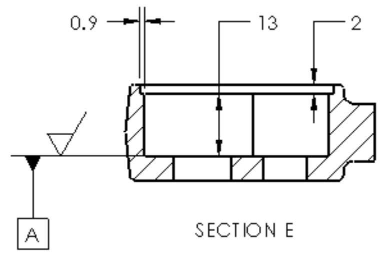
Where to Find It

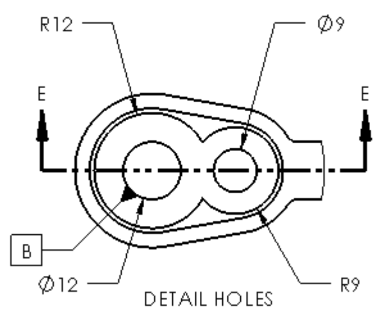
- Click **Surface Finish Symbol** on the Annotation toolbar.

5 Add surface finish.
Click **Surface Finish** and **Machining Required**. Click on the line in `SECTION E` as shown. Click **OK** and drag the symbol along the line.



6 Dimensions.
Use the **Smart Dimension** tool to add the dimensions as shown.





Dimension Properties

The properties of a dimension can be accessed by selecting a dimension. The options are divided into three tabs: **Value**, **Leaders** and **Other**.

Where to Find It

- Click a dimension.

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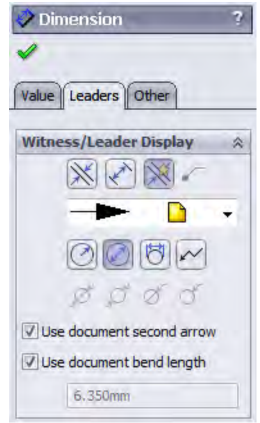
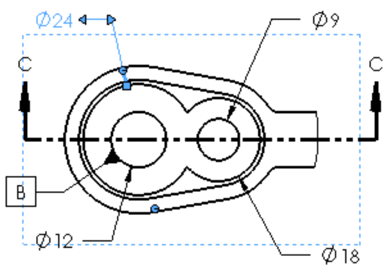
7 Diameters.
Click either of the radial dimensions in DETAIL HOLES. Click the **Leaders** tab of the **Dimension** PropertyManager and click **Diameter**. Repeat for the remaining radial dimension.

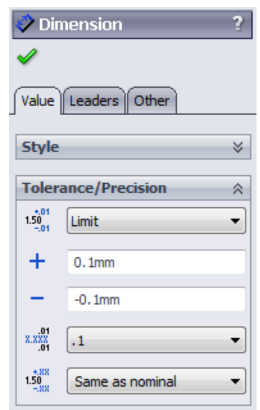


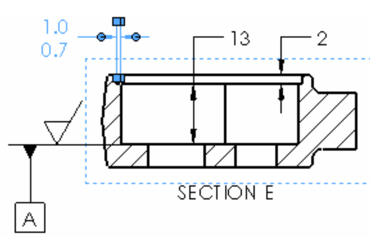


DETAIL HOLES

8 Tolerance.
Click the dimension shown and set the **Tolerance/ Precision** values:

- Tolerance Type = Limit
- Maximum Variation = 0.1
- Minimum Variation = 0.2
- Primary unit precision = .1





Centerlines

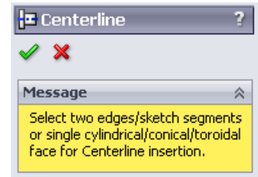
Centerlines are created as fonted lines and arcs in the drawing view.

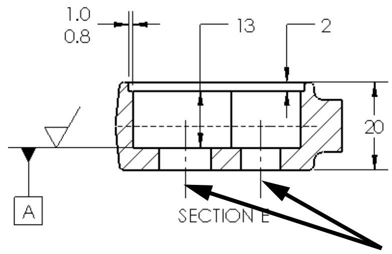
Where to Find It

- Click **Centerline** on the Annotation toolbar.

9 Add centerlines.

Click **Centerline** and select the two cylindrical faces as shown. Also add the **20mm** dimension.





Geometric Tolerance Symbols

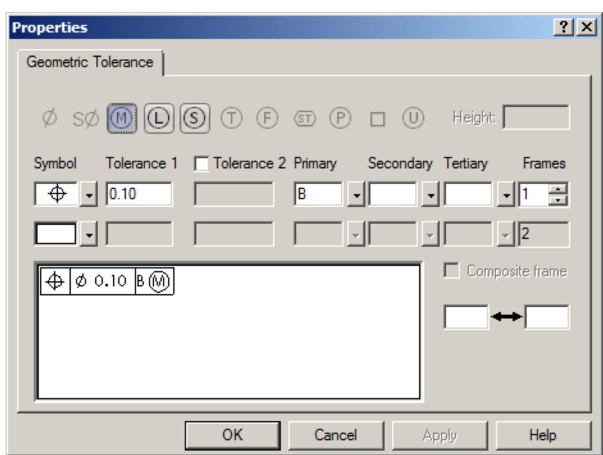
The **Geometric Tolerance Symbol** is used to add geometric tolerances using feature control frames to parts and drawings. SolidWorks supports ANSI Y14.5 Geometric and True Position Tolerancing.

Where to Find It

- Click **Geometric Tolerance Symbol** on the Annotation toolbar.

10 Geometric tolerances.

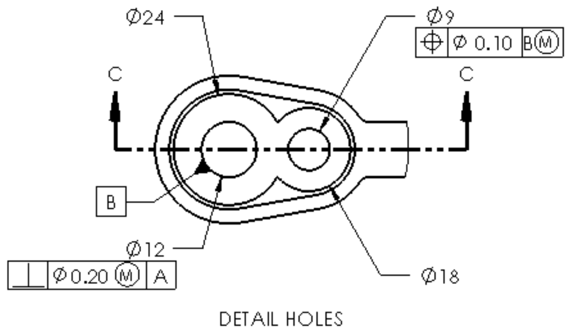
Click **Geometric Tolerance**. Click **No Leader**. Using the symbols, lists and keyboard, create the symbol shown below. Create a second symbol using the same procedure.



Tip

The **Primary, Secondary** and **Tertiary** pulldowns have a combination of type-in and symbol buttons.

11 Drag and drop.
Drag and drop the geometric tolerances onto dimensions as shown. They attach to the dimensions and move with them.

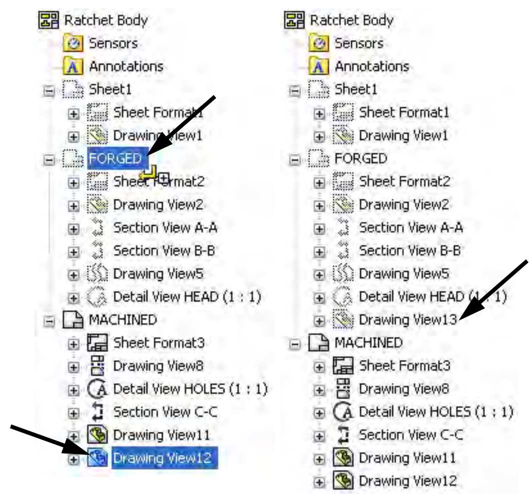


Copying Views

Existing views can be copied and pasted within the same sheet, onto different sheets within the same drawing, or between drawings. They can also be moved without copying.

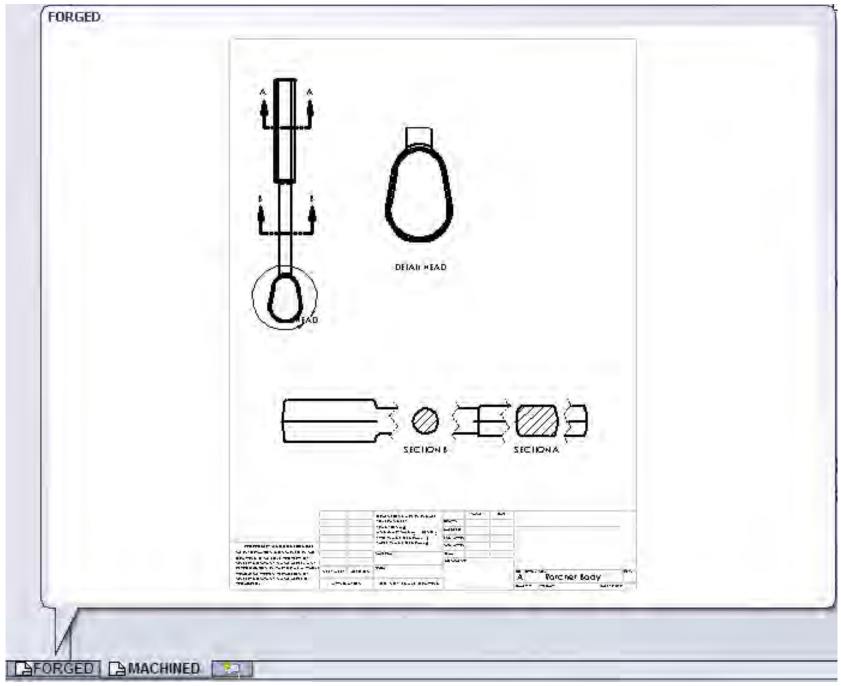
12 Drag and drop.
In the FeatureManager Design Tree, **Control**+drag the last drawing view up and drop it on the `FORGED` sheet.

The copied view is placed on the `FORGED` sheet as a new view.

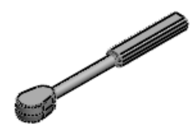


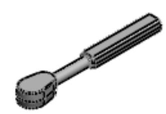
Tip Using drag and drop without the **Control** key moves the view to the drawing sheet.

13 Switch sheets.
Move the cursor over the `FORGED` sheet to see a preview. Click the `FORGED` sheet to open it.

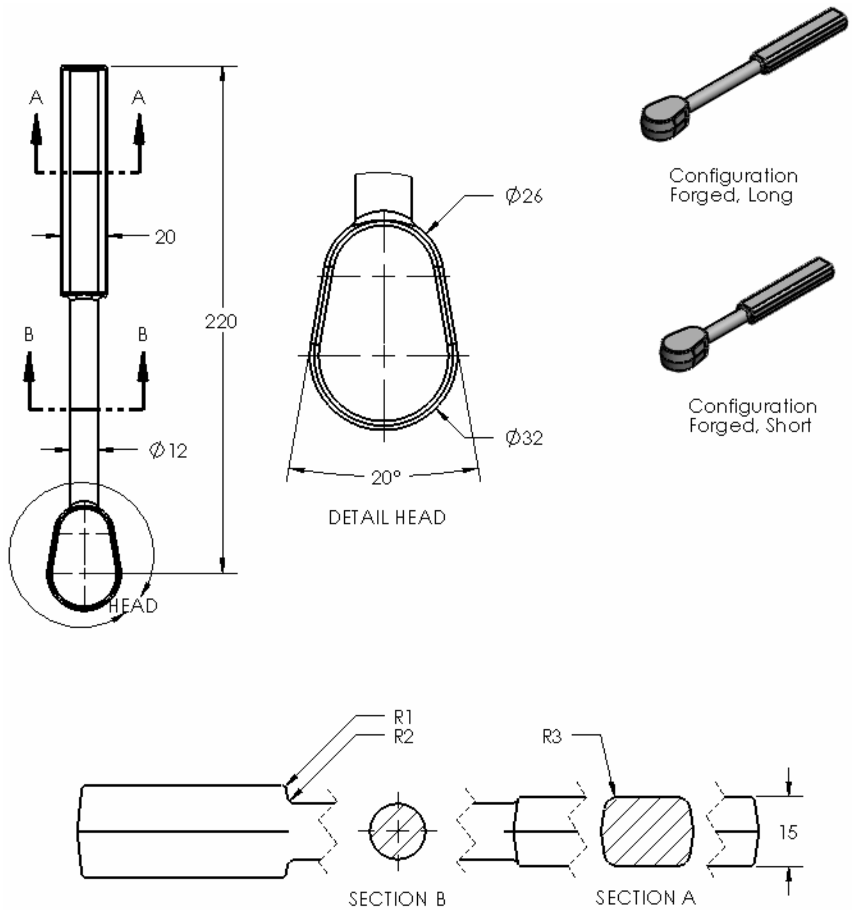


14 Copy in sheet.
Copy the view again and set the configurations for each as shown.



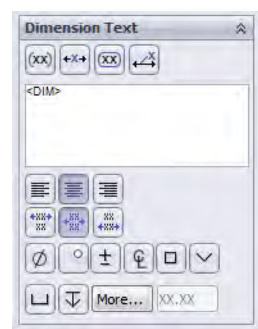


15 Add dimensions and centerlines.
Add dimensions and centerlines to the drawing as shown.



Dimension Text

The **Dimension Text** group box that appears when you select a dimension allows you to prefix, append or replace the text of a dimension. The actual text is shown as <DIM> in the field. Click before or after the text (or use **Enter** to add lines) to place text and/or symbols.



Deleting the <DIM> text eliminates the text of the dimension for replacement.

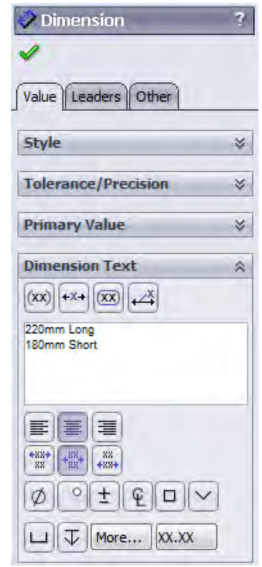
Tip The lower portion of the group box contains commonly used symbols such as diameter, degrees and centerline.

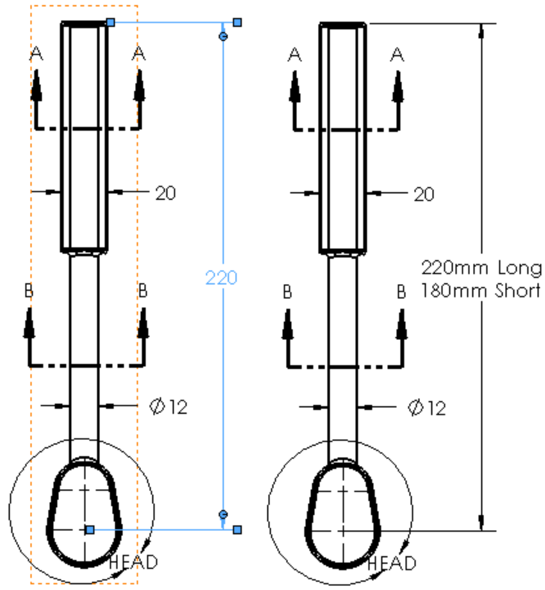
16 Overriding dimension text.
Click the **220mm** dimension and delete the <DIM> text. This message appears: `Overriding the dimension value text <DIM> disables tolerance display. Do you want to continue?` Click **Yes**.

17 Text.
Type this text and click **OK**.

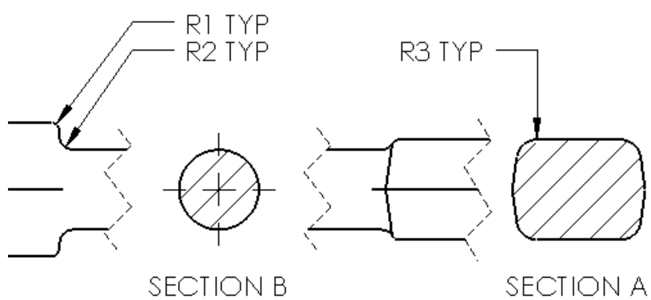
220mm Long

180mm Short





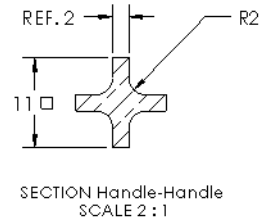
18 Append dimension text.
Select all three of the radial dimensions in the view. Place the cursor after the dimension text (R<DIM>), press the **Space** bar and type TYP. Click **OK**.



19 Save and close the files.

Exercise 52: Details and Sections

Create a multiple sheet drawing of a part.



This lab reinforces the following skills:

- *Section View* on page 379.
- *Model Views* on page 382.
- *Broken View* on page 383.
- *Projected Views* on page 387.
- *Copying Views* on page 394.
- *Dimension Text* on page 396.

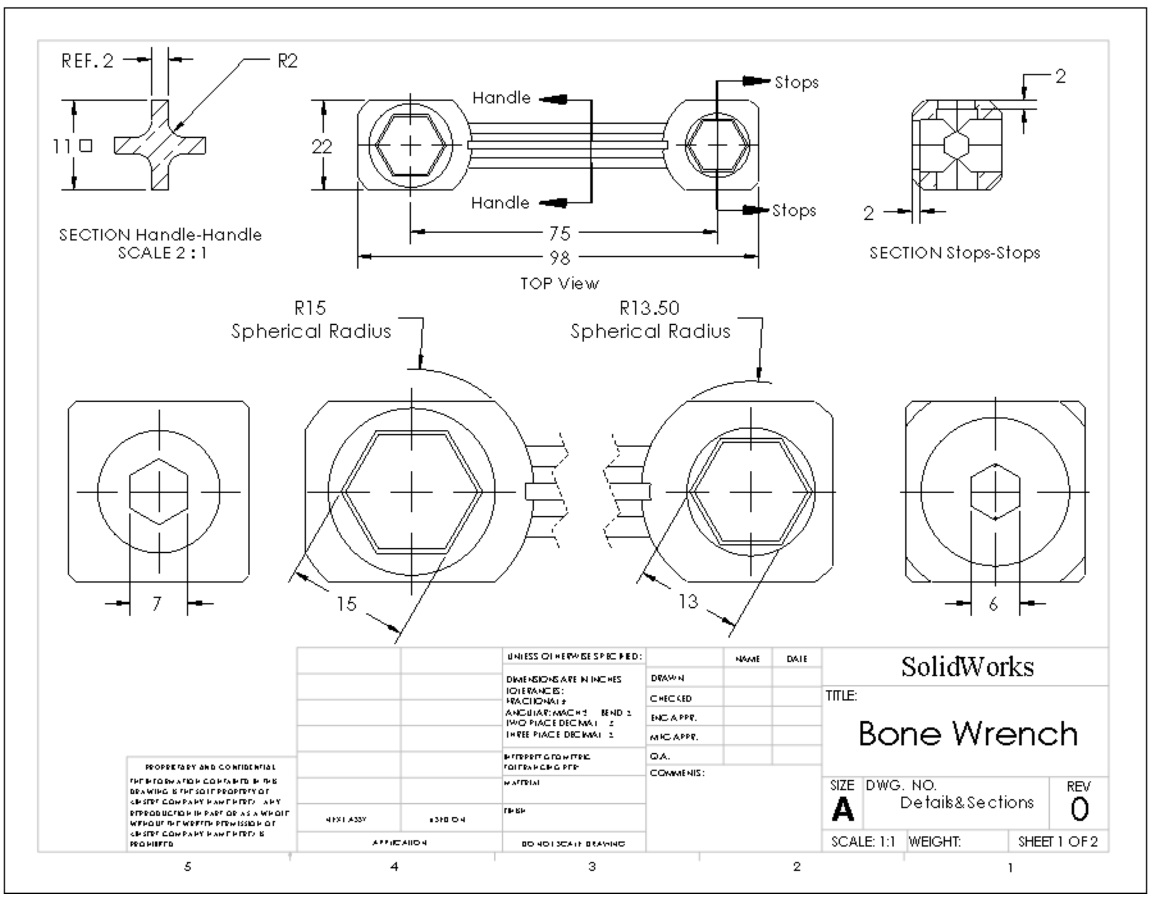
Units: **millimeters**

Procedure

Use the part Details&Sections to create drawings shown here and on the following page.

Sheet1

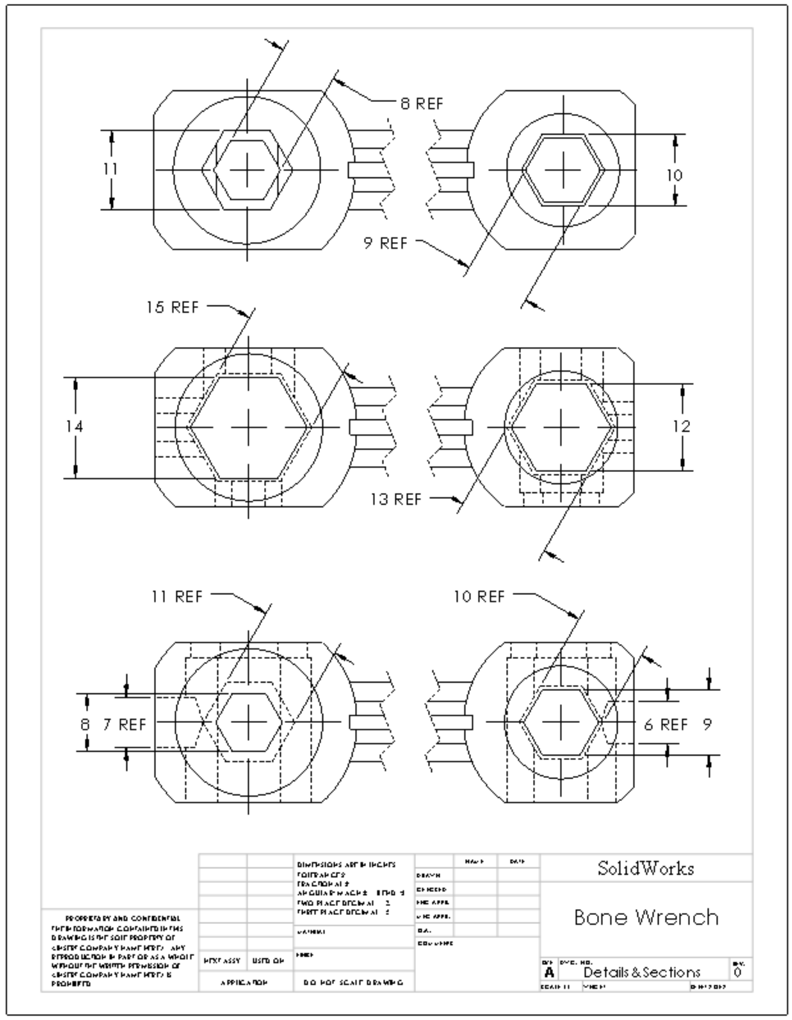
Create the drawing shown below using an **A (ANSI) Landscape** template.



Note

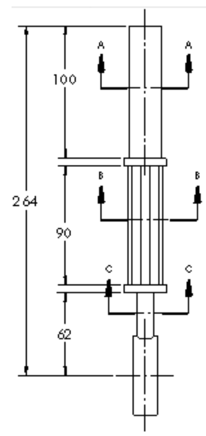
The view labelled Top View and the broken view below it are both oriented to the Top view.

Sheet2 Create the drawing shown below using an **A (ANSI) Portrait** template.



Exercise 53: Broken Views and Sections

Create a multiple view drawing of a part.



This lab reinforces the following skills:

- *Section View* on page 379.
- *Model Views* on page 382.
- *Broken View* on page 383.
- *Datum Feature Symbols* on page 390.
- *Centerlines* on page 393.
- *Dimension Properties* on page 392.
- *Dimension Text* on page 396.

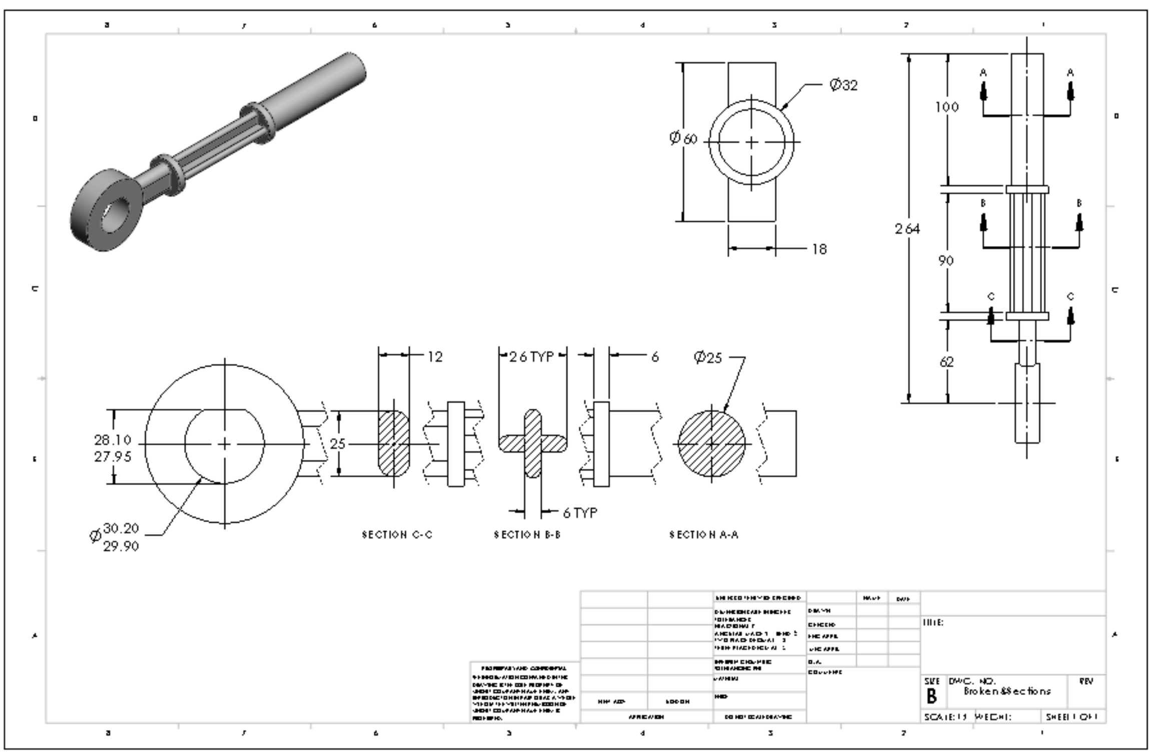
Units: **millimeters**

Procedure

Use the part Broken&Sections to create drawings shown here and on the following page.

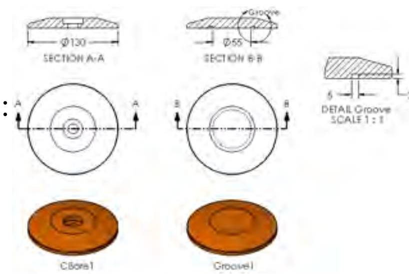
Sheet1

Create the drawing shown below using an **B (ANSI) Landscape** template.



Exercise 54: Drawings

Create a multiple view drawing of a part with configurations.



This lab reinforces the following skills:

- *Section View* on page 379.
- *Detail Views* on page 385.
- *Notes* on page 389.

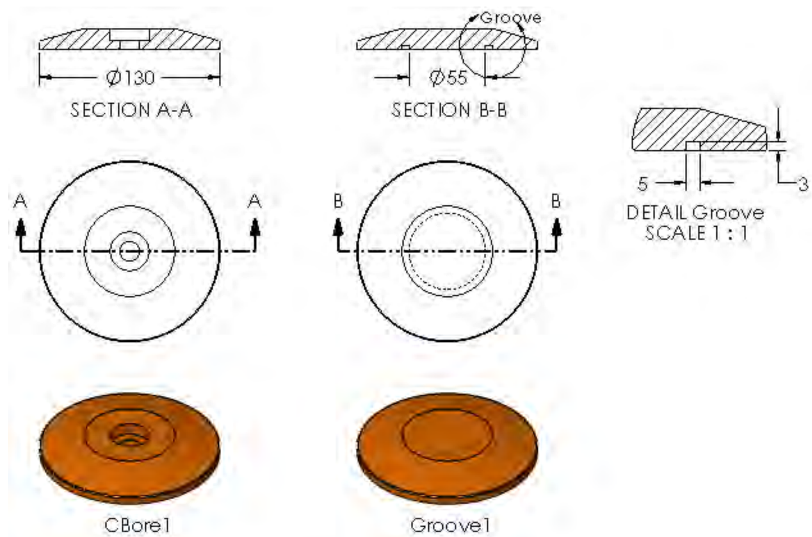
Units: **millimeters**

Procedure

Use the part Design for Configs to create a B-size drawing similar to the one shown here. See details below.

Views

Model and section views.



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.

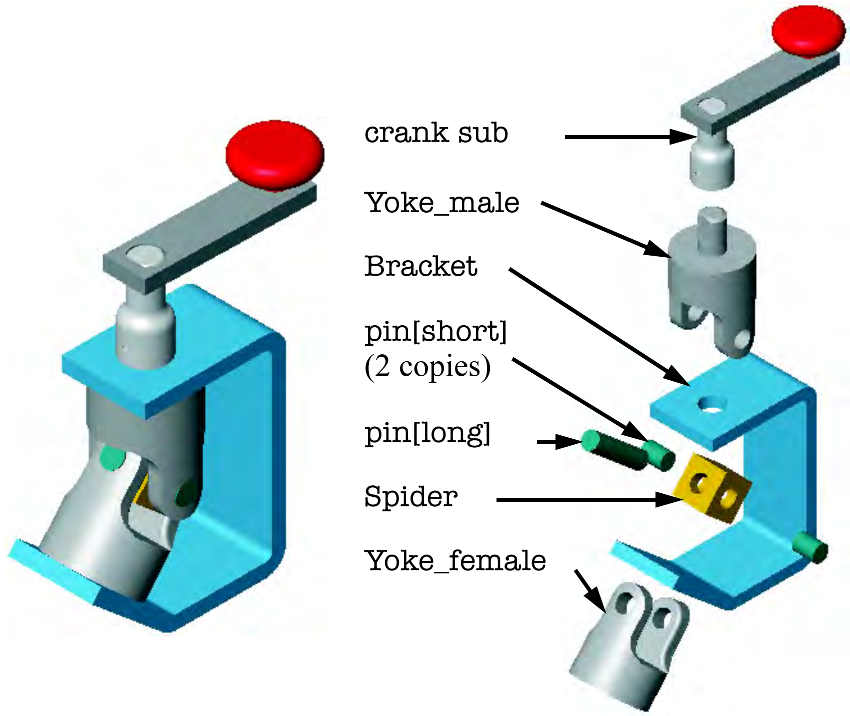
Stages in the Process

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 design tree 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.

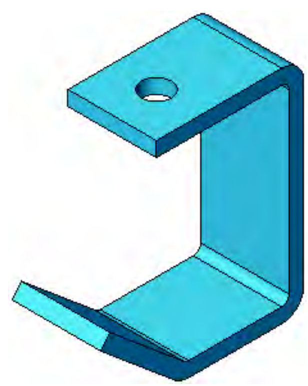
The Assembly

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 planes and a `Mates` folder.

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.

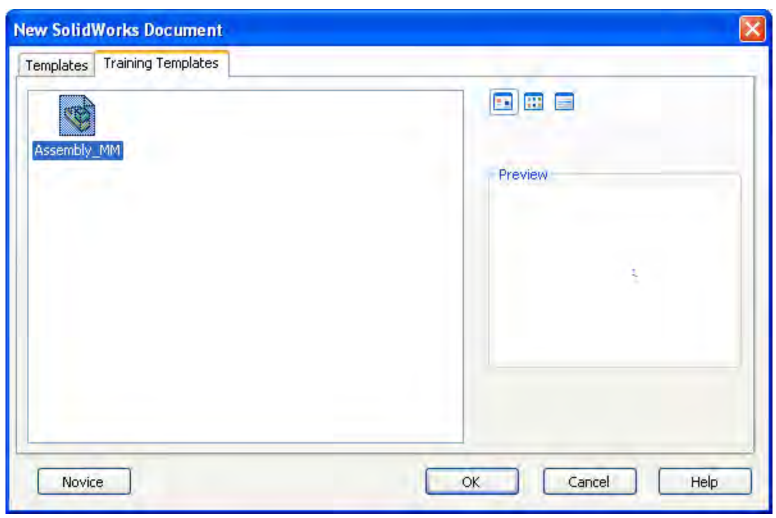
Where to Find It

- 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_MM`.



Shortcut

Double-click the desired template to automatically open a new assembly document using that template.

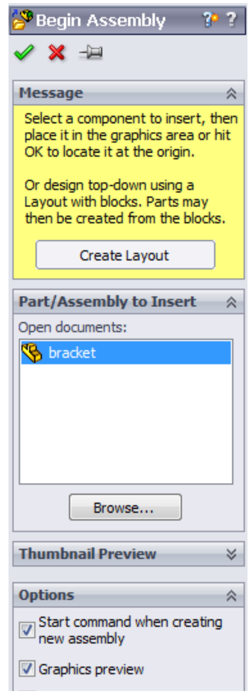
Note

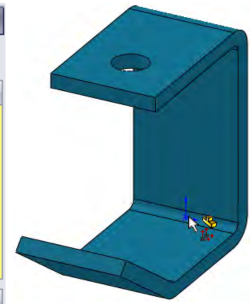
The units of the assembly can be different from the units of the parts. For example, you can assemble a mixture of inch and millimeter parts in an assembly whose units are feet. However, when you edit the dimensions of *any* of the parts in the context of the assembly, they will be displayed in the units 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 by placing the cursor at the origin or by simply clicking **OK**.

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 planes of the component match the planes of the assembly, and the component is fully defined.

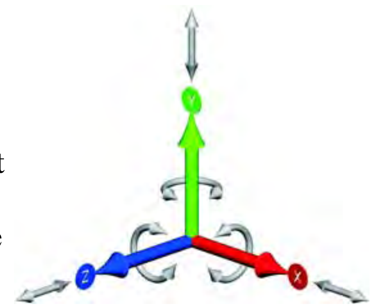
Consider assembling a washing machine. The first component logically would be the frame onto which everything else is mounted. By aligning this component with the assembly's planes, we would establish what could be called "product space". Automotive manufacturers refer to this as "vehicle space". This 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 a part is listed there, they will be described.

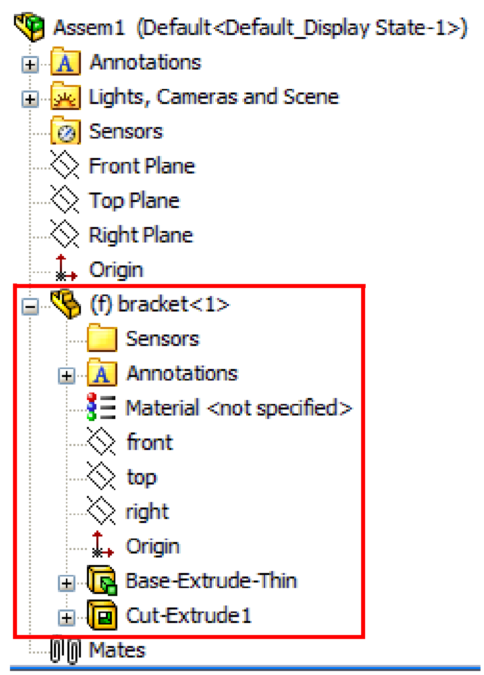
Degrees of Freedom

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 is used internally to distinguish each instance of the component from each other when multiple instances of the component are included in the assembly.

Tip

Instances are not renumbered for deletions. The highest instance number may not reflect the total.

- **Component Part Folder**
  Each component part contains the entire contents of the part, including all features, planes and axes.

External Reference Search Order

When any parent document is opened, all documents that are referenced by the parent document are also loaded into memory. In the case of assemblies, components are loaded in memory according to the suppression state they were in when the assembly was saved.

The SolidWorks software searches for referenced documents in the following order:

1. **Random Access Memory**
   If a file with the correct name is already in memory, SolidWorks will use that file.
2. **The directory paths specified in the Folders list on the File Locations tab in Tools, Options (optional)**
   Users may establish a list of directories for SolidWorks to search first. Generally, these directories would be shared network locations where projects are stored. Establishing this list is optional and may be bypassed.
3. **The last path you specified to open a document**
   When you open a parent document, SolidWorks will search in the same directory for the referenced files.
4. **The last path the system used to open a document**
   This applies if the system opened a referenced document last.
5. **The path where the referenced document was located when the parent document was last saved**
   This is the path stored in the parent document except that the drive path (C:\, D:\, etc.) is considered to be the current drive.
6. **The path where the referenced document was located when the parent document was last saved with the original disk drive designation**
   This is the absolute path name stored with the parent document.
7. **If a referenced file is still not found, SolidWorks gives you the option to browse for it**
   When SolidWorks reaches the end of the search list and has not found a document, it turns the process over to the user to search manually.

Note

All updated reference paths in the parent document are saved when you save the parent document.

File Names

File names should be *unique* to avoid bad references. If you have two different parts called `bracket.sldprt`, a parent document that is looking for the part will find whichever comes first in the search order.

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**

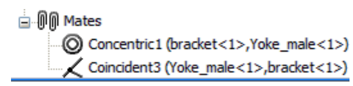
Any features below the marker are suppressed. Individual components cannot be rolled back.

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 partially 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.**
- **Drag them from the Task Pane.**

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.

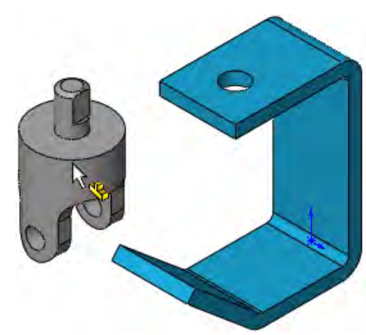
Where to Find It

- Click **Existing Part/Assembly** on the Assembly toolbar.
- Or, click **Insert, Component, Existing Part/Assembly...**.

---

5 Insert `Yoke_male`.

Click **Insert, Component, Existing Part/ Assembly...** and select the `Yoke_male` part using the **Browse...** button. Position the component on the screen to the left of the `bracket` 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.

Tip

Clicking on a component in the FeatureManager design tree will cause that component to highlight. Also, moving the cursor to a component in the graphics window will display the feature name.

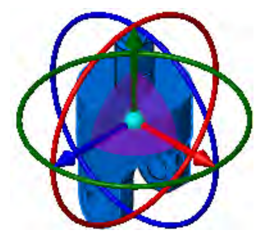
Moving and Rotating Components

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

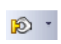
Using the mouse:

- Drag and drop a component.
- Right-click a component, and select **Move with Triad**. The triad provides visible axes, webs (plane between axes) and rings. Use the triad to move or rotate components along axes/ webs or around rings. Float over arrowhead: left-drag to move along the axis. Float over ring: left-drag to rotate around the ring.



Using the menus:

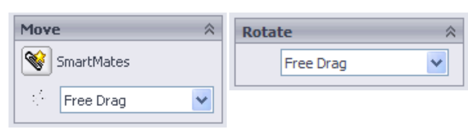
- From the pull-down menu choose: **Tools, Component, Rotate** or **Move**.
- Right-click the component, and select **Move...**.
- Or, on the Assembly toolbar pick one of these tools:

 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.

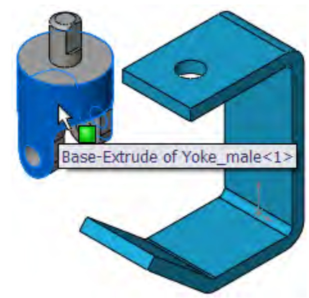


The **Move** tool has several options for defining the type of movement. The option **Along Entity** has a selection box, **Along Assembly XYZ**, **By Delta XYZ**, and **To XYZ Position** require coordinate values.

The **Rotate** tool also has options to define how the component will rotate.

6 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.



Mating Components

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 *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

Most mates are made between a *pair* of objects.

Where to Find It

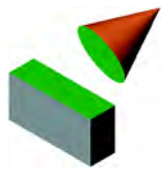
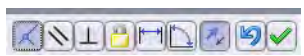
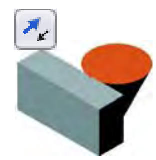
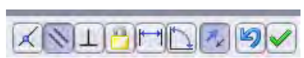
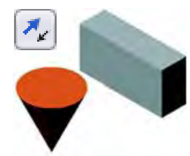
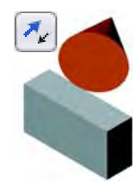
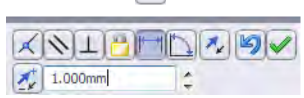
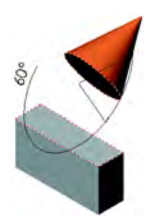
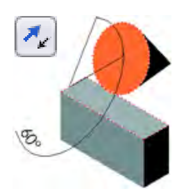
- On the **Insert** menu, select **Mate...**.
- Or, on the Assembly toolbar, click **Mate**.
- Or, right-click a component and choose **Mate**.

Note

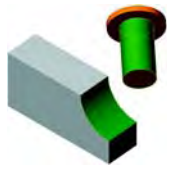
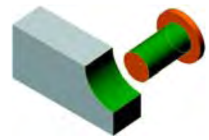
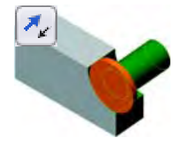
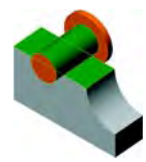
Mates have icons that are based on their type, for example **Coincident**.

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  		
Lock Select anywhere on component. 	Components that are locked will move together. No alignment options.	

Common Buttons

There are three buttons common to all the controls:

- is **Undo**
- is **Flip Mate Alignment**
- is **OK** or **Add/Finish Mate**

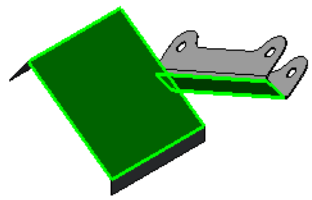
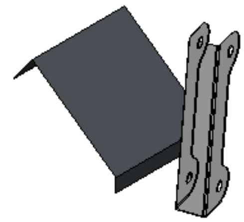
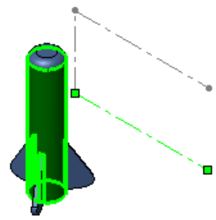
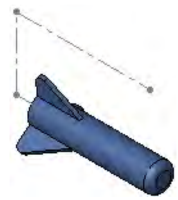
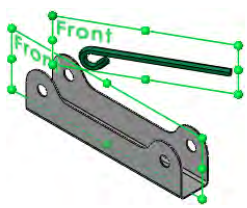
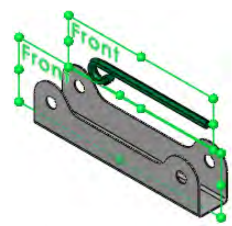
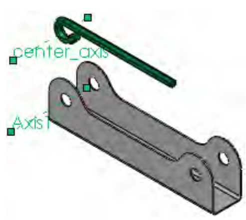
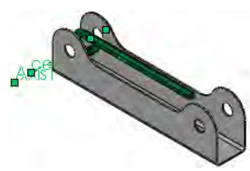
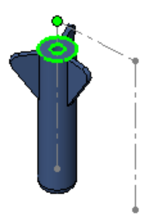
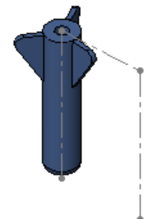
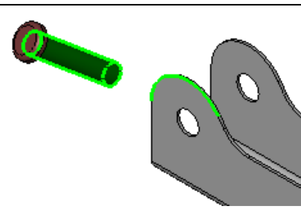
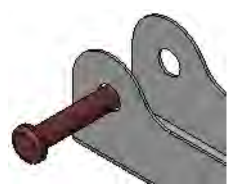
In addition to these, the **Mate** dialog itself also has specific mate alignment controls, and .

Tip

After the mate has been created, you can right-click and select **Flip Mate Alignment** to reverse the alignment.

Things to which you can mate

There are many types of topology and geometry that can be used in mating. The selections can create many mate types.

Topology/ Geometry	Selections	Mate
Faces or Surface		
Line or Linear Edge		
Plane		
Axis or Temporary Axis		
Point, Vertex, Origin or Coordinate System		
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 design tree. 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.

Selection Filter

The selection filter option is very useful in mating. Since many mates require face selections, the filter can be set to allow selection of only faces. Filters remain in effect until SolidWorks or the part is exited, or the filter is changed or cleared.

Where to Find It

- Click **Toggle Selection Filter Toolbar** on the Standard toolbar and select one or more filter types.
- Press the **F5** key.

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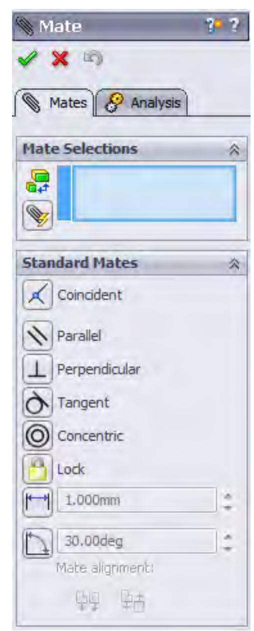
7 Selection filter.

Toggle the **Selection Filter Toolbar** on and set the **Select** option to faces.

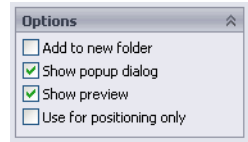
8 Mate PropertyManager.

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 414.

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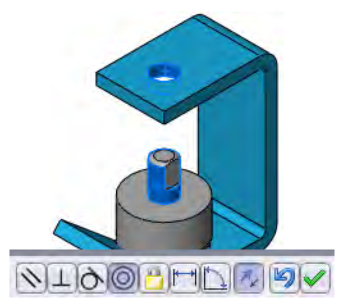
9 Selections and preview.
Select the faces of the Yoke_male and the bracket as indicated.



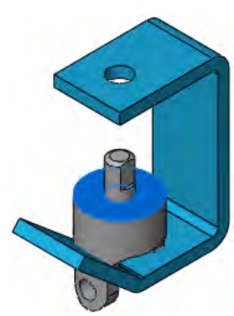
As the second face is selected, the **Mate** pop-up toolbar is displayed.

Concentric is selected as the default and the mate is previewed.

10 Add a mate.
The faces are listed in the **Mate Settings** list. Exactly two items should appear in the list.

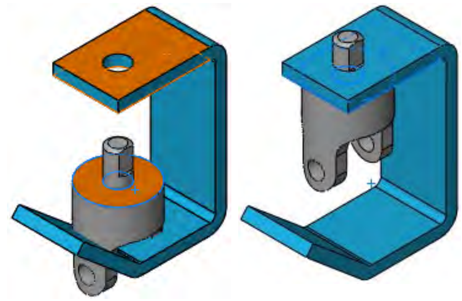
Accept the **Concentric** mate and click **Add/Finish Mate** (check mark).

11 Planar face.
Select the top planar face of the Yoke_male component.

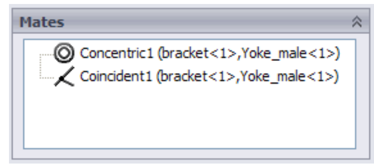


12 Select Other.

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.



13 Mates listed.

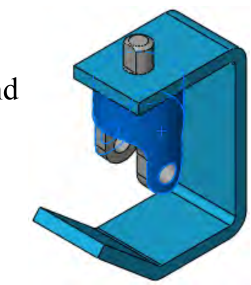


The mates, concentric and coincident, remain listed in the **Mates** group box. They will be added to the Mates folder when the **OK** button on the PropertyManager dialog is clicked. They can also be removed from this group box so that they are *not* added. Click **OK**.

14 State of constraint.

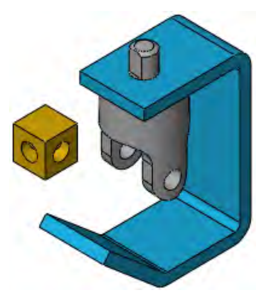
The Yoke_male component is listed as under constrained. It is still able to move by rotating around the axis of its cylindrical surface.

Test the behavior of the Yoke_male by dragging it.



15 Add the spider.

Use **Insert Component** to add the spider component.

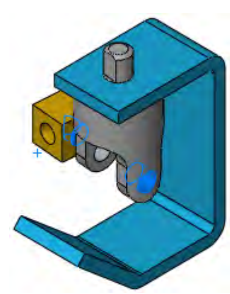


16 Concentric mate for spider.

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 one of the **Advanced Mates** from 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

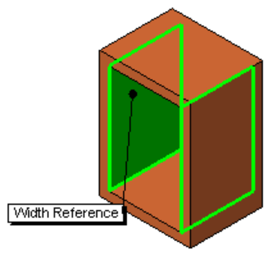
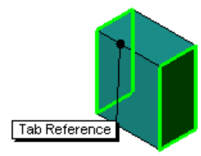
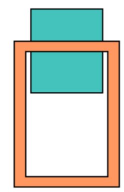
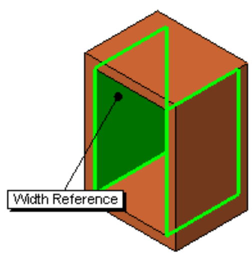
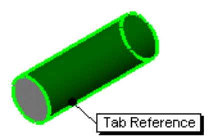
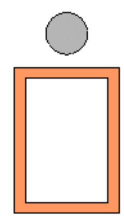
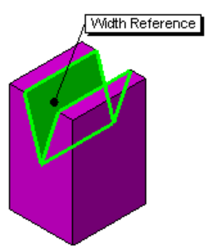
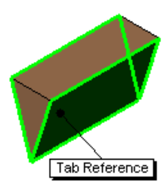
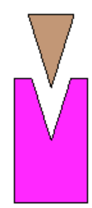
The remaining advanced mates are discussed in the *Assembly Modeling* manual.

Width References

The **Width** selections form the “outer” faces, used to contain the other component.

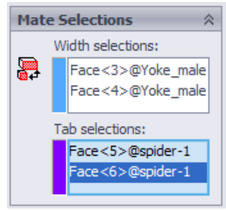
Tab References

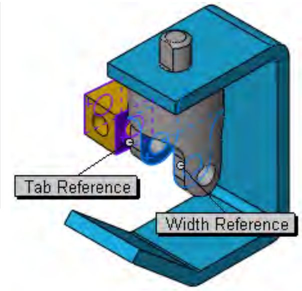
The **Tab** selection(s) form the “inner” faces, used to locate the component.

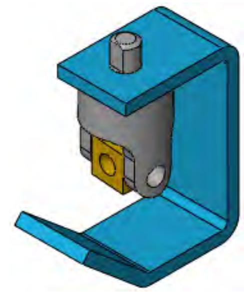
Width selections	Tab selection(s)	Result
		 (Front view)
	 (single selection)	 (Front view)
		 (Front view)

17 Width mate.
Click **Insert, Mate** and select the **Advanced Mates** tab.

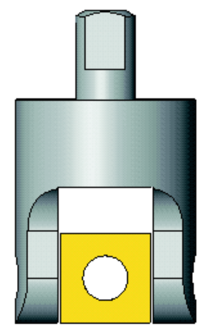
Click the **Width** mate and select the **Width selections** and **Tab selections** as shown.





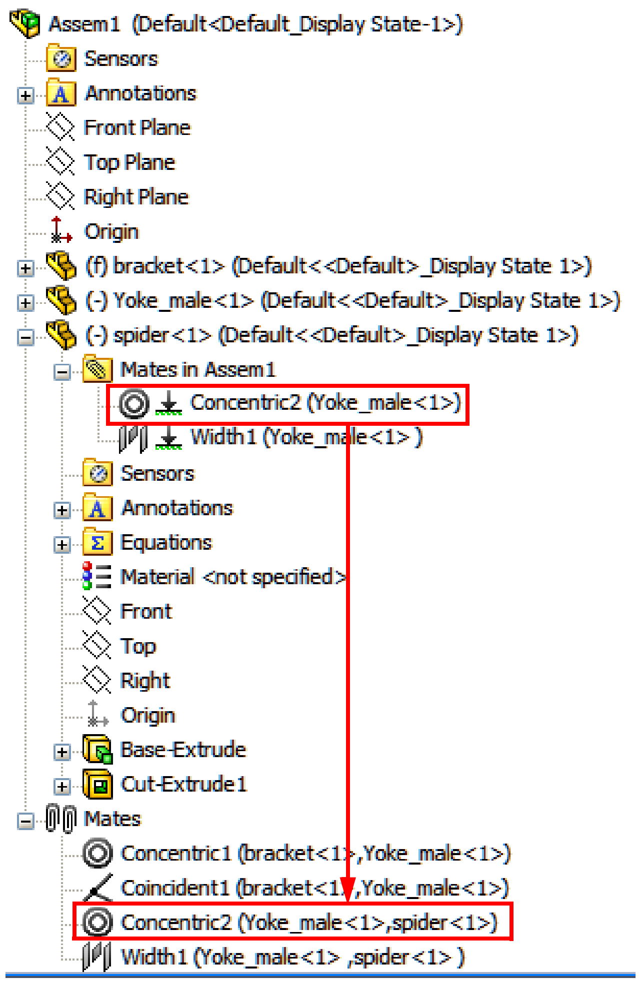


18 Results.
The mate keeps the `spider` centered inside the `Yoke_male` with equal gaps on each side.



19 Mates by component.

Expand the spider component in the FeatureManager design tree. 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.

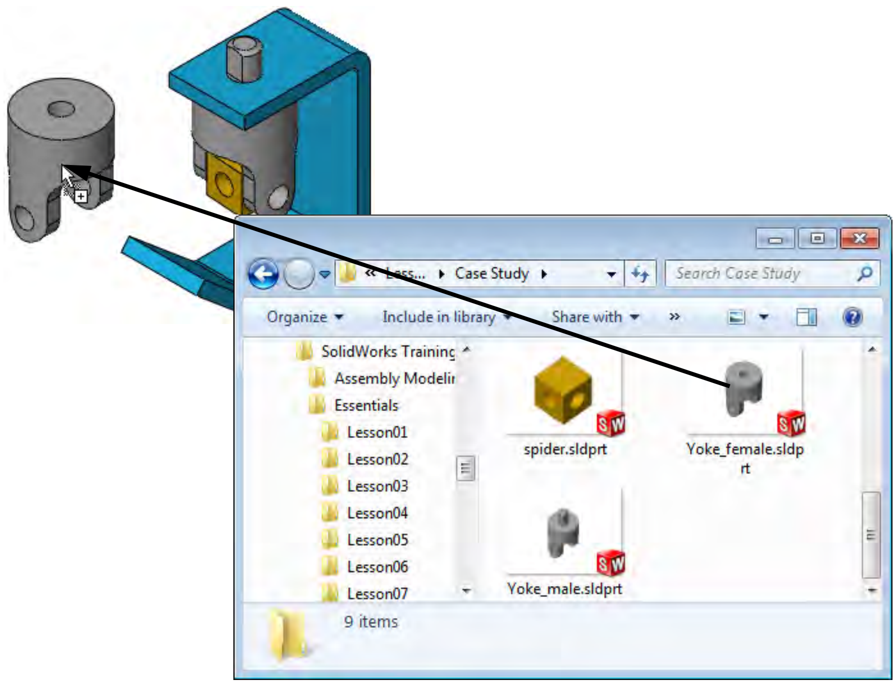
Note

The icon indicates that the mate is in the path to ground, or, it is one of the mates that keeps the component in position.

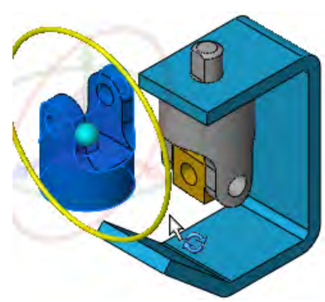
Adding Components Using Windows Explorer

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.

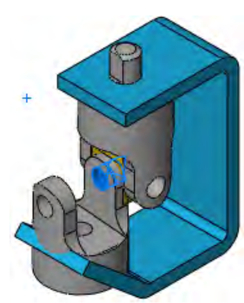
20 Open Windows Explorer.
Size the Windows Explorer window so the SolidWorks graphic area can be seen. Since SolidWorks is a native Windows application, it supports standard Windows techniques like "drag and drop". The part files can be dragged from the Explorer window into the assembly to add them. Drag and drop the `Yoke_female` into the graphics area.



21 Rotate using Triad.
Right-click the `Yoke_Female` and select **Move with Triad**. Drag the ring as shown.

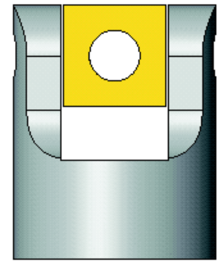


22 Concentric mate.
Select the cylindrical faces as shown and add a **Concentric** mate between them.



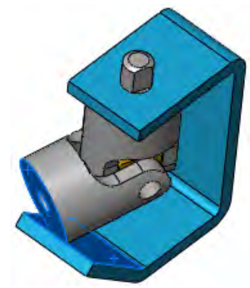
23 Second width mate.
Add a **Width** mate between the spider and the Yoke_female.

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.

The **Coincident** mate is selected as default and a warning dialog appears:

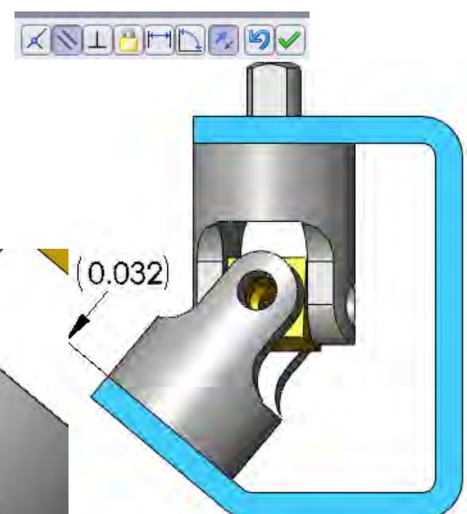


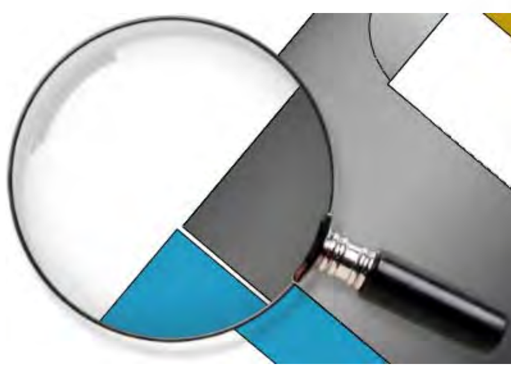
The default mate type (coincident) would over-define the assembly. Please select the mate type below.

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. Press **g** to use the magnifying glass and view the gap.





Dynamic Assembly Motion

Drag under defined components to display the motion allowed by the remaining degrees of freedom.

Note

Components that are fixed or fully defined cannot be dragged.

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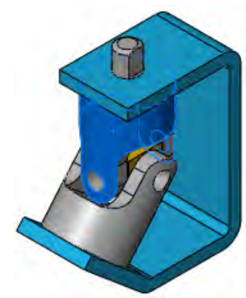
26 Drag components.

Drag the Yoke_male component to turn it. The mated components spider and Yoke_female move with it.



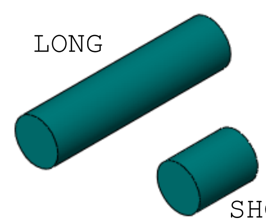
Displaying Part Configurations in an Assembly

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.

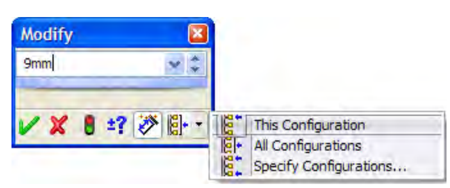


Using Part Configurations in Assemblies

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 several ways to create this type of configuration within a part:

- Applying different dimension values to individual configurations as shown at the right.
- Use Modify Configurations.
- Design tables.



Drag and Drop from an Open Document

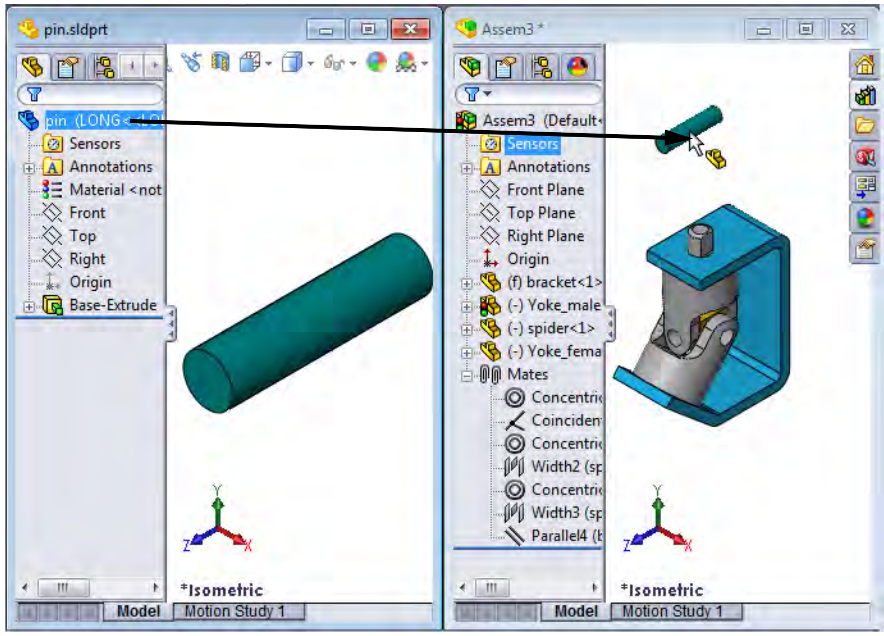
The pin will be inserted by dragging it in from an open document window into the assembly.

Note

If the bracket window is still open, close it before the next step.

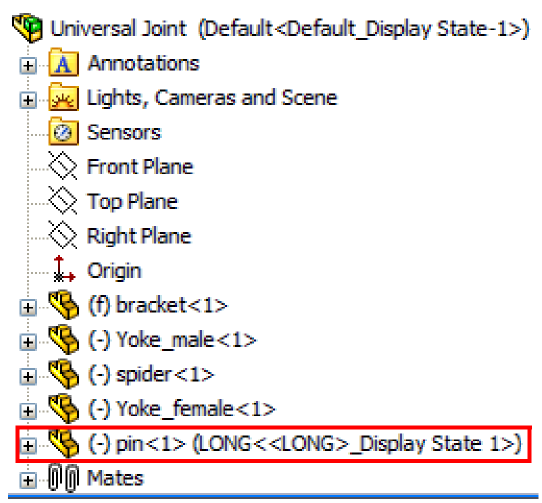
27 Drag and drop.

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 design tree. An instance of the pin is added to the assembly.



Important!

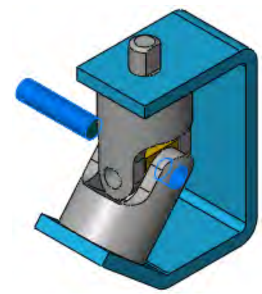
The pin is a component that contains multiple configurations. Components like this display the configuration and display state they are using as part of the component name. In this case the configuration used by instance <1> is LONG. The display state is Display State 1 within the configuration <LONG>. Each instance can use a different configuration/display state combination.



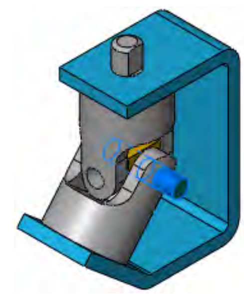
Note

Display States are primarily used in assemblies, but can be used in multi-body parts. For more information on display states, see the *Assembly Modeling* training manual.

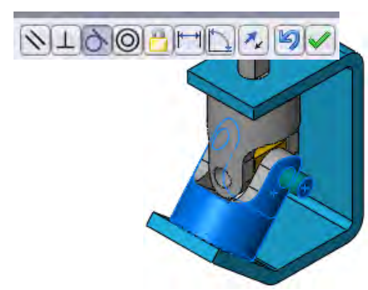
28 Concentric mate.
Add a **Concentric** mate between the cylindrical face in the Yoke_female and pin.



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

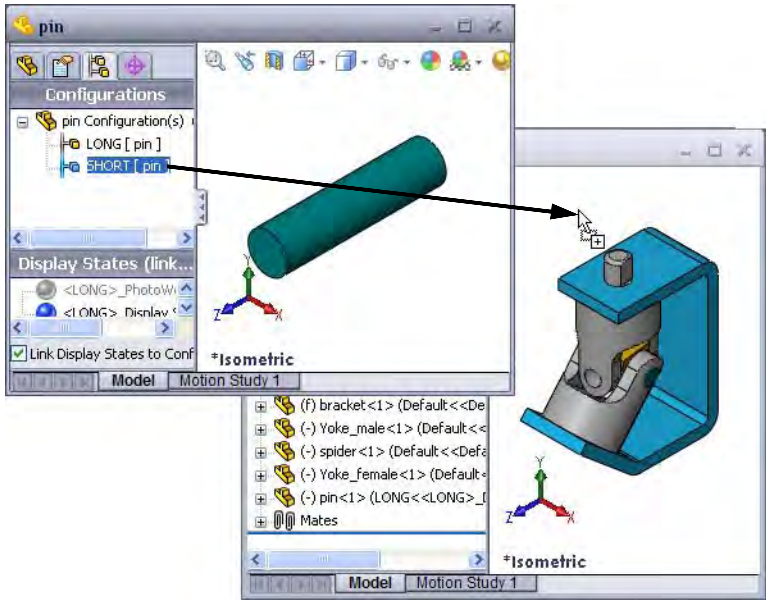
When you need to access a component while working in an assembly, you can open it directly, without having to use the **File, Open** menu. The component can be either a part or a sub-assembly.

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.

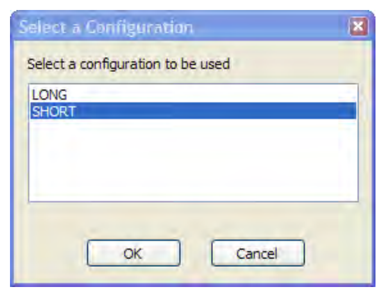
Drag and drop the configuration SHORT into the graphics window of the assembly. You can drag and drop *any* configuration from the ConfigurationManager, not just the active one.



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.



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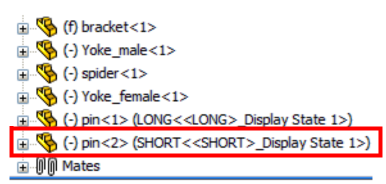
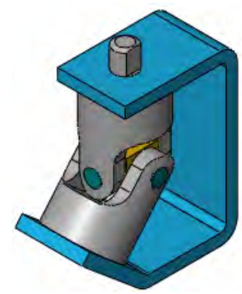
32 Second instance.

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.

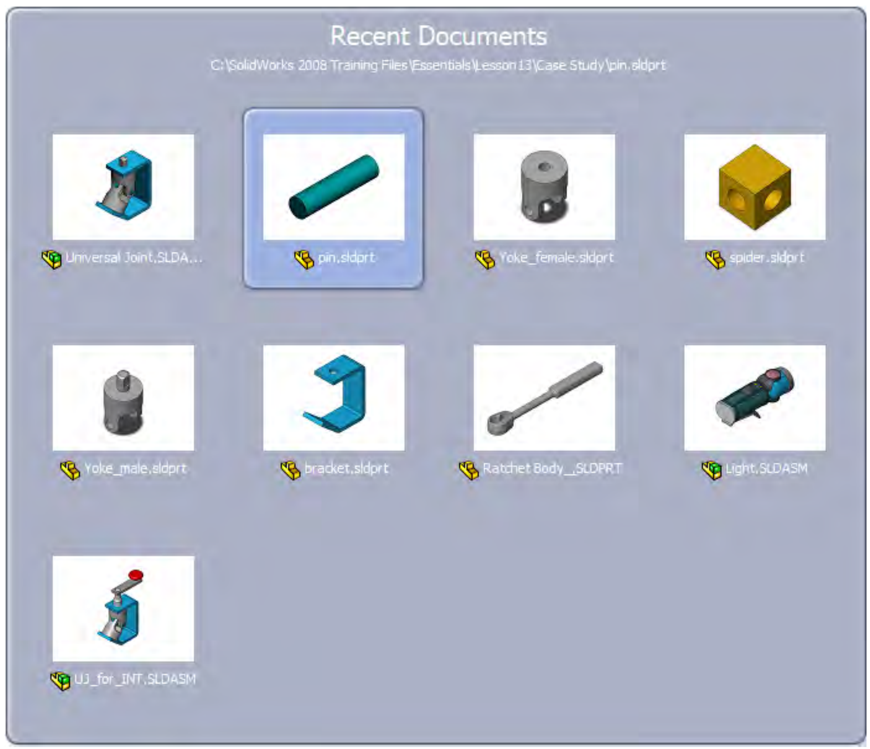


33 Mate the component.
Add **Concentric** and **Tangent** mates to mate the second instance of the pin.



Recent Documents

SolidWorks maintains a list of recently opened documents that can be used to access documents quickly. Type the shortcut key **R** and click the document to open.



Where to Find It

- Type the shortcut **R**.

Tip

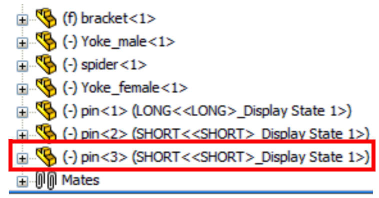
The full pathname of the selected document is listed at the top of the screen similar to **Ctrl+Tab**.

34 Switch documents.
Switch to the **pin.sldprt** document, close it and maximize the assembly window.

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.

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 and dropping it in the graphics area. The result is another instance that uses the SHORT configuration, since it was copied from a component with that configuration.

Tip

You can also drag a copy by selecting the component in the graphics area.

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> .

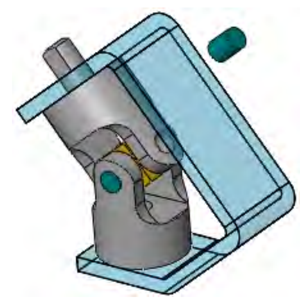
Show Component turns the display back on.

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 components** or **Show components**.
- Right-click the component and select **Component Properties...**. 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 design tree 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**.

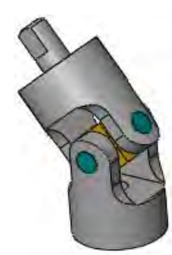
---

36 Hide the bracket.
Change the view orientation from the default Isometric by pressing **Shift+Left Arrow** once. Click on the bracket component and hide it using the **Hide/Show Component** tool.

Important!

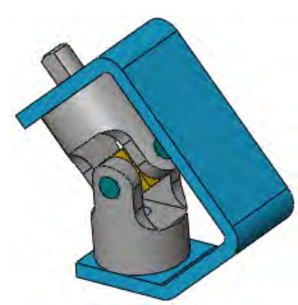
Use **Hide Component** *not* **Hide Solid Body**. **Hide Solid Body** will hide the solid within the part.

37 Complete the mating.
Complete the mating of this component by adding **Concentric** and **Tangent** mates using **Insert Mate**.



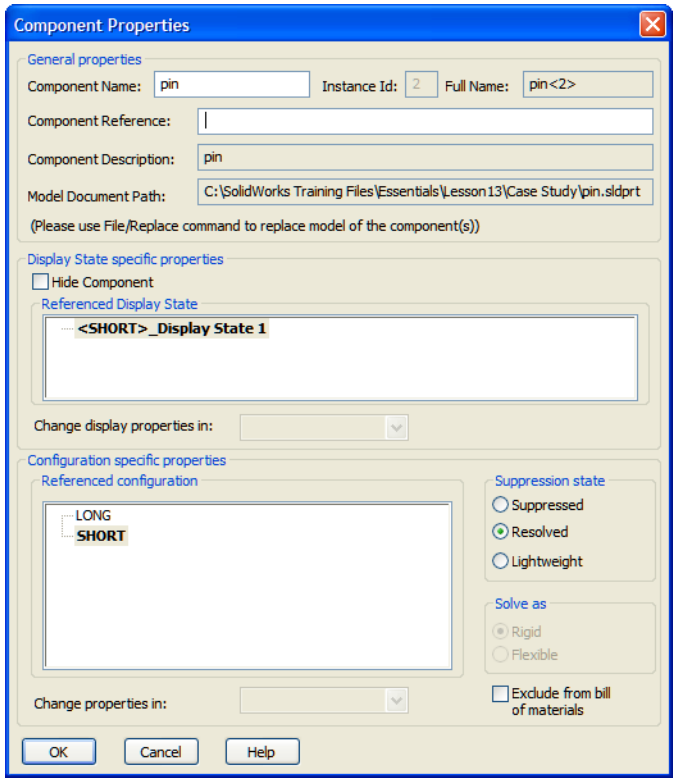
38 Show the component.
Select the bracket again and click **Hide/Show Component** to toggle the graphics back on.

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 instance references with a different file, use **File, Replace...**.
- **Display State specific properties**
  Hides or shows the component. Also enables you to select a display state by name.
- **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.

Where to Find It

- Click or right-click a component and select **Component Properties** .

40 Component properties.

Click the pin<3> component and select **Component Properties**. The **Referenced configuration** option is set to SHORT. This dialog box can be used to change the configuration, suppress, or hide an instance. 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 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.

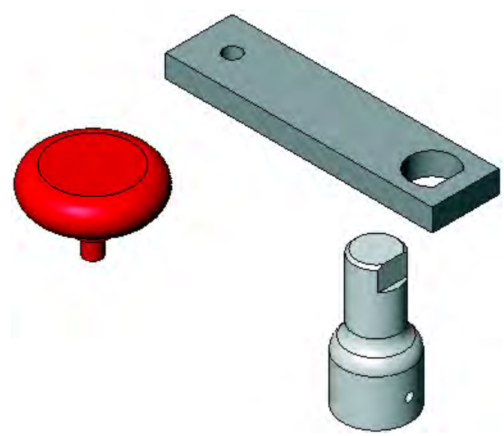
Create a new assembly using the Assembly_MM template.

Click **Keep Visible** on the **Begin Assembly** PropertyManager and add the crank-shaft component. Locate it at the origin of the assembly. It is **Fixed**.

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.

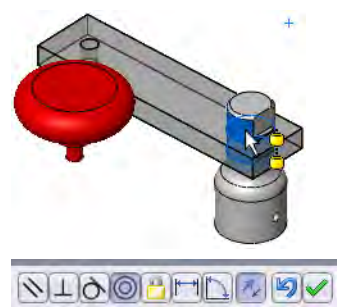
These mates use the same **Mate** pop-up toolbar as the **Mate** tool uses to set the type and other attributes. All mate types can be created with this method.

Certain techniques generate multiple mates and do not use the toolbar. These require the use of the **Tab** key to switch mate alignment.

---

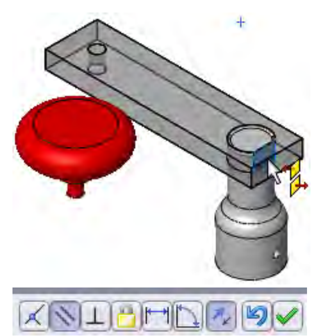
43 Smart Mate concentric.
Follow these steps to add a **Concentric** mate through the **Smart Mate** technique:

1. Press and hold the **Alt** key.
2. Click and hold the circular face of the `crank-arm`.
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.



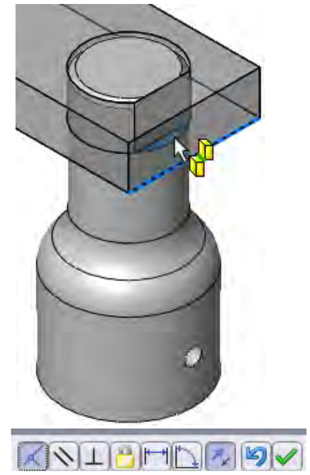
A **Concentric** mate is added between the `crank-arm` and the `crank-shaft` components.

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.

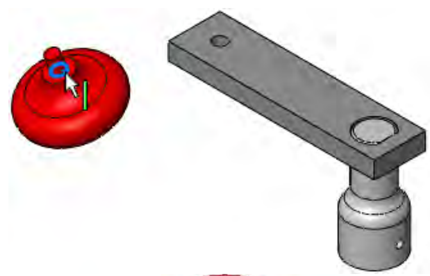


Use the **Mate** pop-up toolbar to *switch* to a **Parallel** mate.

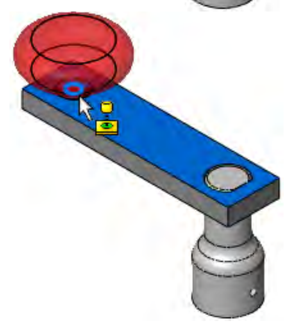
45 Coincident.
Select the *edge* of the `crank-arm` and **Alt-+drag** it to the flat on the `crank-shaft`. Drop the component when the symbol appears, indicating a **Coincident** mate between and edge and a planar face. Use the **Mate** pop-up toolbar to confirm the **Coincident** mate.



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.

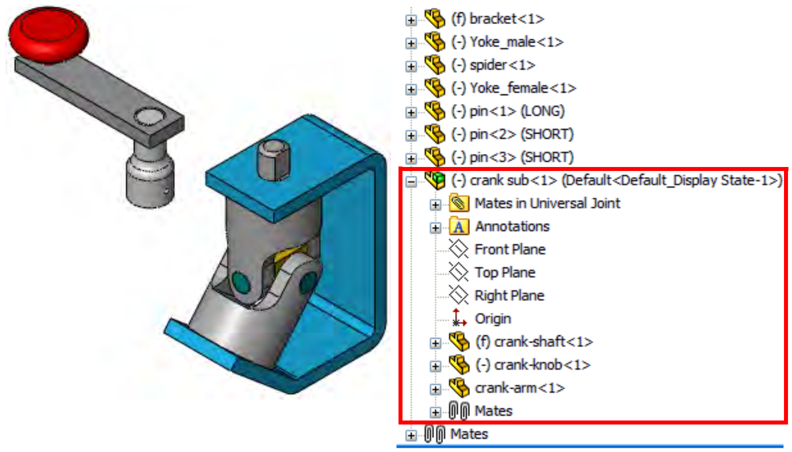
47 Save.
Save the assembly, naming it `crank sub`. Leave the assembly open.

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.
Switch to the main 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.

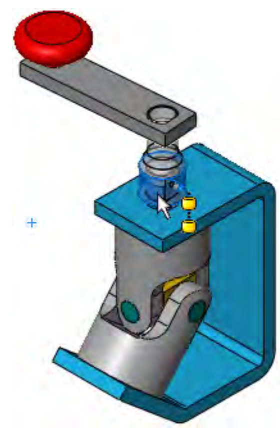
49 Place the sub-assembly.
Place the sub-assembly near the top of the Yoke_male component. Expanding the sub-assembly component icon shows all the component parts within it, including its own mate group.



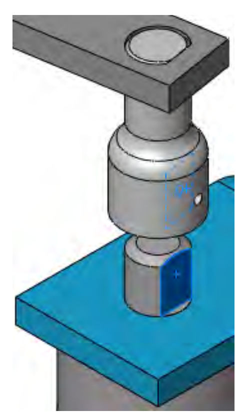
Mating Sub-assemblies

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.

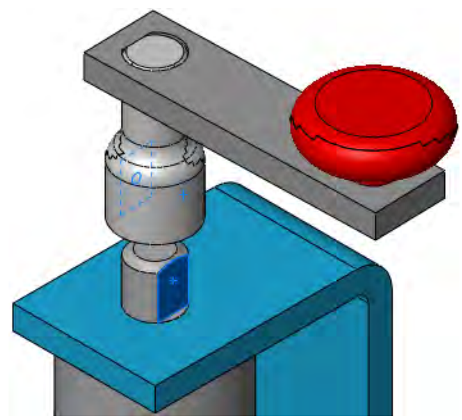
50 Smart Mate concentric.
Add a **Concentric** mate, using **Alt+drag**, between the cylindrical surfaces of the post on the top of the Yoke_male and the crank-shaft.



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.



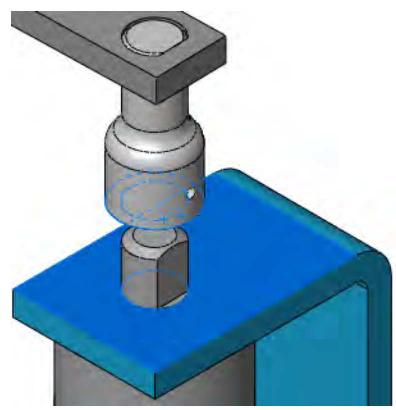
Question: Why wouldn't you use a **Coincident** mate here?

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.

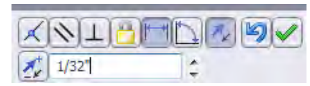
Distance Mates

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 which 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 in units that are different than the current units. Type **1/32”**.



Note

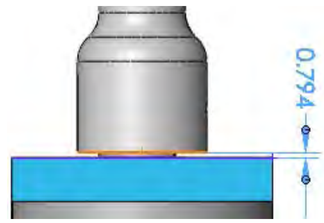
You could type **in** or **“** after the number. The system will automatically convert it to the current units, millimeters.

If the `crank-shaft` penetrates into the `bracket` select the **Flip Dimension** button.

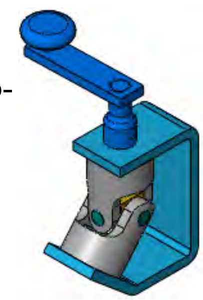
Click **OK** to create the mate.

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 millimeters.



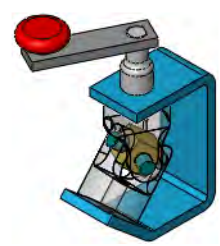
55 Select in the FeatureManager design tree.
Select the sub-assembly `crank sub` in the FeatureManager design tree. All components in the sub-assembly will be selected and highlighted.



Tip

From the graphic window, right-click a component of the sub-assembly and click **Select Sub-assembly**.

56 Dynamic Assembly Motion.
Use **Change Transparency** on the yokes. Drag the `crank-arm` to see the motion of the `spider`.

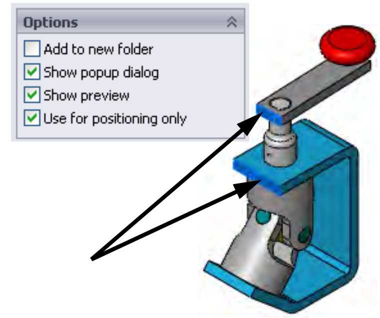


Use For Positioning Only

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. Save the assembly.



Pack and Go

Pack and Go is used to collect and copy all the files used by the assembly into a single folder or zip file. It is especially useful when the entire assembly must be sent to another user and the files are stored in many different folders.

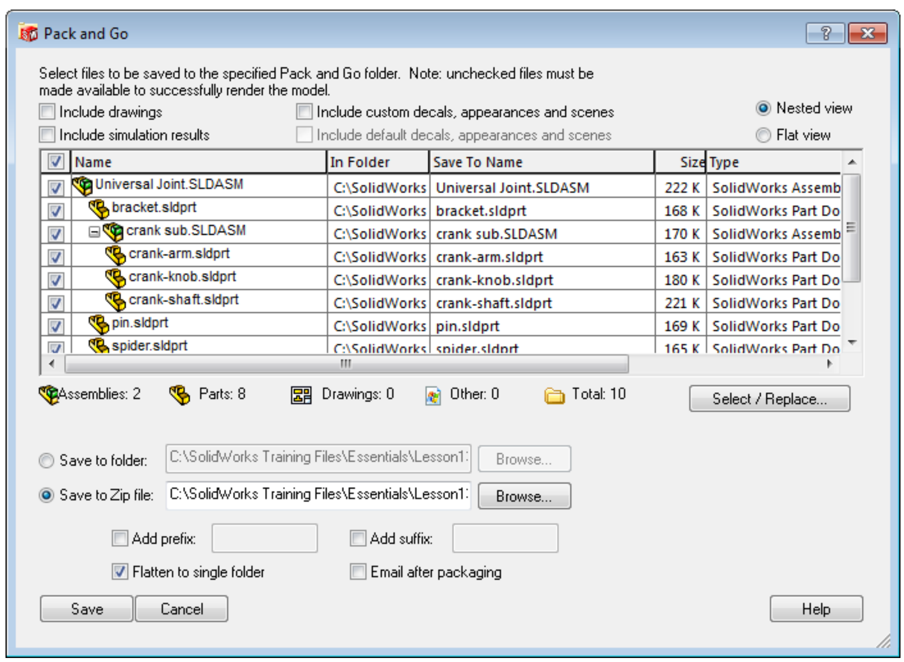
Note

Drawings and Simulation results can also be collected and copied.

Where to Find It

- From the pull-down menu, choose **File, Pack and Go**.

58 Pack and Go.
Click **File, Pack and Go** and select **Save To Zip File** using the default name and **Flatten to single folder**.

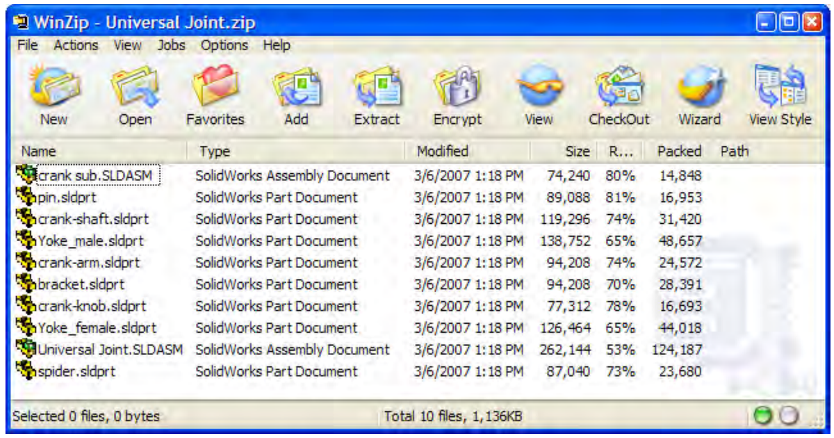


Click **Save**.

59 Save and close the part.

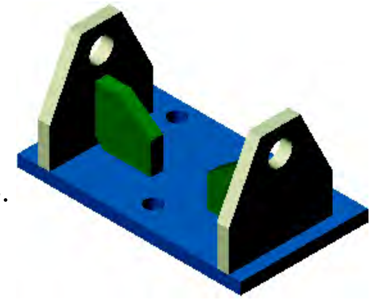
60 Zip file.

Double-click the zip file Universal Joint.zip in the lesson folder.



Exercise 55: Mates

Create this assembly by adding components to a new assembly and using **Insert Mate**.



This lab uses the following skills:

- *Creating a New Assembly* on page 406.
- *Adding Components* on page 411.
- *Mating Components* on page 413.

Units: **millimeters**

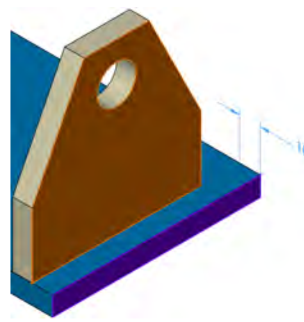
Procedure

Create a new assembly.

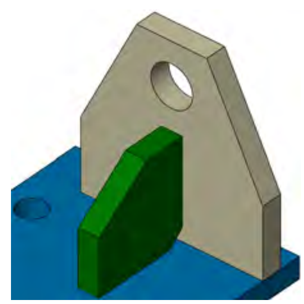
1 **Add the component Base.**
Create a new assembly.

Drag the RectBase from the Mates folder into the assembly and fully constrain it to the assembly origin.

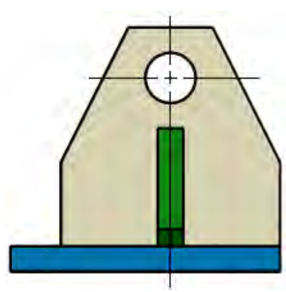
2 **Add the EndConnect.**
Add an instance of the EndConnect to the assembly. **Mate** it to the RectBase using a distance of **10mm** and two coincident mates as shown.



3 **Add the Brace.**
Add an instance of the Brace to the assembly. **Mate** it to the RectBase using coincident mates as shown.



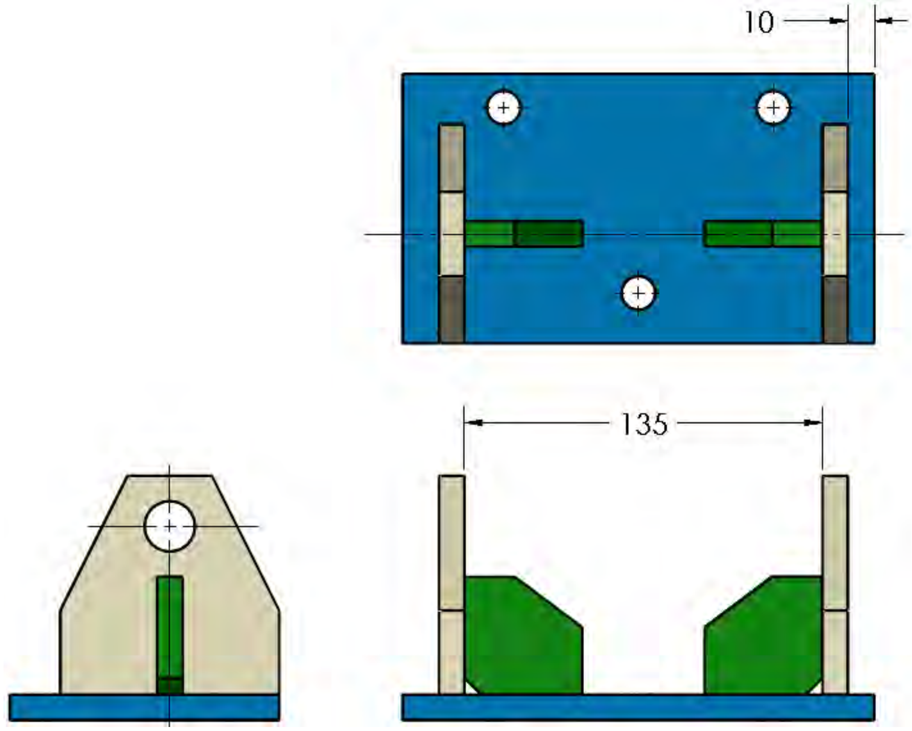
The Brace is centered on the hole in the EndConnect component.



Tip

Coincident mates between planes can be used to center components.

4 Additional components.
Add more instances of the Brace and EndConnect components, placing them as shown.



5 Save and close all files.

Exercise 56: Gripe Grinder

Assemble this device by following the steps as shown.

This lab uses the following skills:

- *Creating a New Assembly* on page 406.
- *Adding Components* on page 411.
- *Mating Components* on page 413.
- *Dynamic Assembly Motion* on page 425.

Units: **millimeters**

Procedure

Create a new assembly.

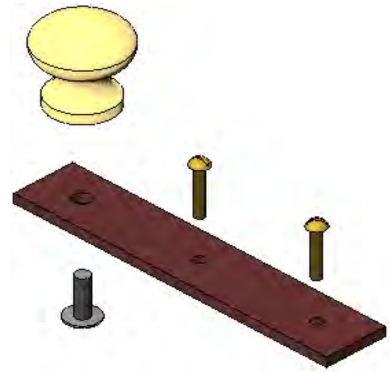
1 Add the component Base.
Create a new assembly.

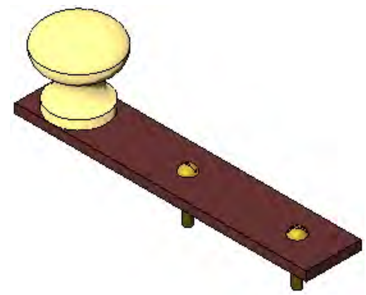
Drag the Base from the Grinder Assy folder into the assembly and fully constrain it to the assembly origin.

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_MM 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.

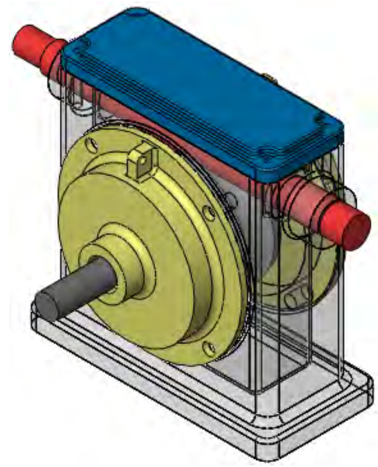
6 Mate the Crank assembly to the Sliders.
The two RH Machine Screws go into the holes in the Sliders. The underside of the Handle mates to the top face of one of the Sliders.

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.

8 Save and close the assembly and the part.

Exercise 57: Using Hide and Show Component

Create this assembly by using mates only. No dimensions are provided.



This lab uses the following skills:

- *Creating a New Assembly* on page 406.
- *Adding Components* on page 411.
- *Mating Components* on page 413.
- *Component Hiding and Transparency* on page 430.
- *Smart Mates* on page 434.

Units: **millimeters**

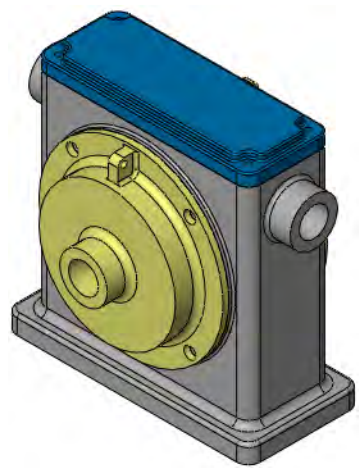
Procedure

Create a new assembly.

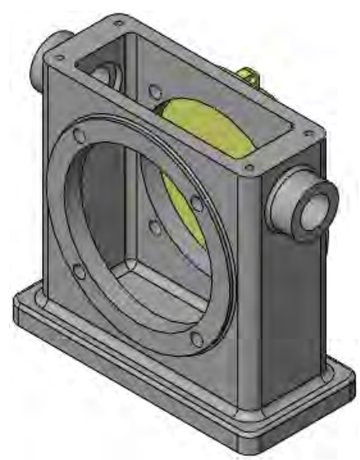
1 **Create assembly.**
Open the Housing component found in the Gearbox Assy folder. Use **Make Assembly from Part/Assembly** to create a new assembly with the Assembly_MM template.

2 **Add the components.**
Drag or insert the remaining component parts into the assembly.

3 **Mates.**
Mate the Cover Plate and both Cover_Pl&Lug components to the Housing as shown.



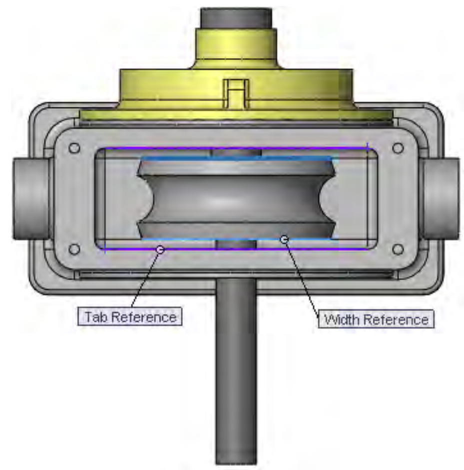
4 **Hide.**
Hide the Cover Plate and one of the Cover_Pl&Lug components as shown.



5 **Add more components.**
Add the Worm Gear Shaft and Worm Gear components as shown.

Tip

Mate the Worm Gear to the Housing using a **Width** mate.



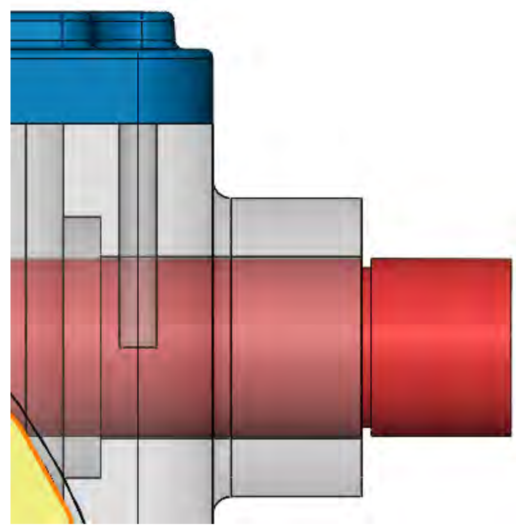
6 Detail.

Show the hidden components. Use **Change Transparency** to change the appearance of the Housing.

Add the Offset Shaft component and mate it.

Tip

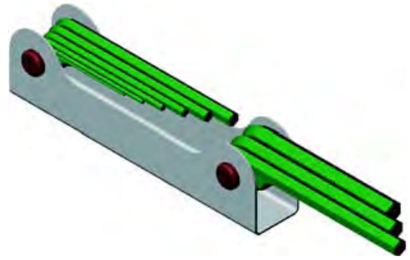
A detail for mating the Offset Shaft to the Housing is shown at right.



7 Save and close all files.

Exercise 58: Part Configurations 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.



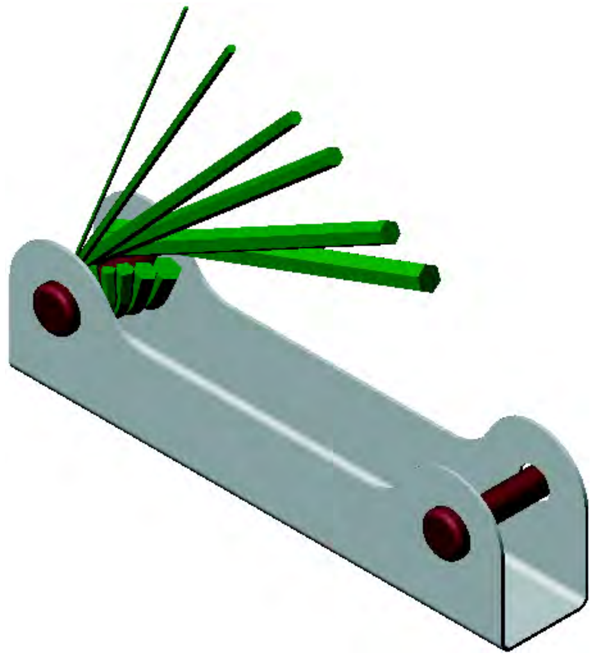
This lab reinforces the following skills:

- *Adding Components* on page 411.
- *Mating Components* on page 413.
- *Using Part Configurations in Assemblies* on page 425.
- *Dynamic Assembly Motion* on page 425.
- *Opening a Component* on page 427.

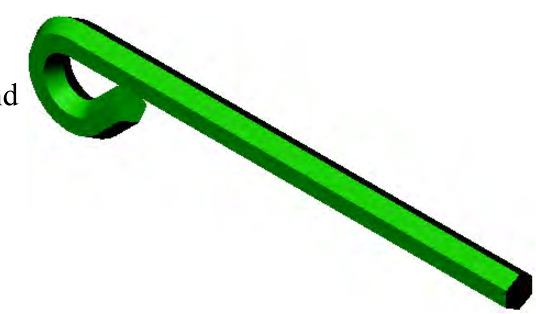
Procedure

Open an existing assembly.

1 **Existing assembly.**
Open the assembly `part configs` found in the `part configs` folder. 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.



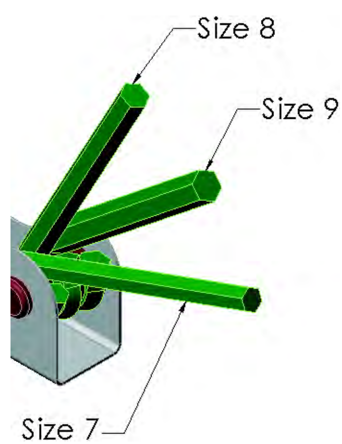
3 Configuration.

Edit the table LENGTH under the Tables folder. Change the values in the Length column.

	Length
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.

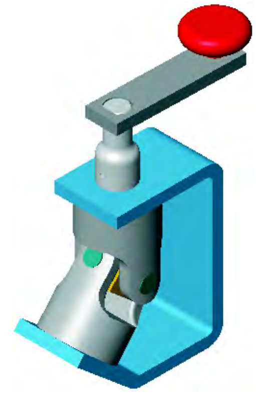
5 Save and close the assembly and the part.

Exercise 59: U-Joint Changes

Make changes to the assembly created in the previous lesson.

This exercise uses the following skills:

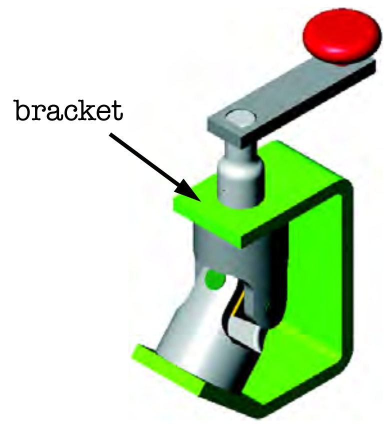
- *Insert Component* on page 411.
- *Mating Components* on page 413.
- *Opening a Component* on page 427.
- *Component Hiding and Transparency* on page 430.



Procedure

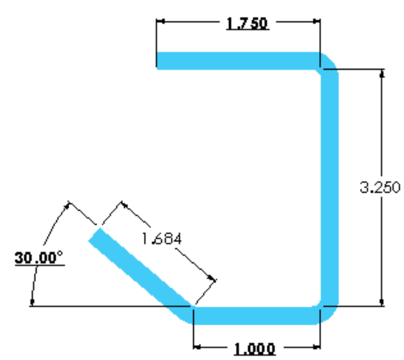
Open an existing assembly.

1 **Open the assembly named `Changes`.**
Open the assembly `Changes` found in the `U-Joint Changes` folder.



2 **Open the `bracket` component.**
From the FeatureManager design tree or the screen, open the component `bracket<1>` for editing.

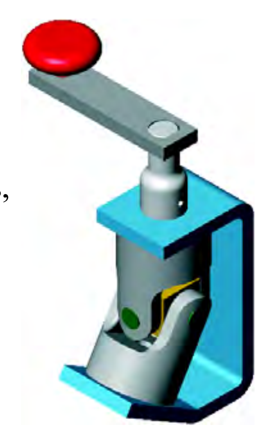
3 **Changes.**
Double-click the first feature and change the dimensions that are shown as bold and underlined.



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.

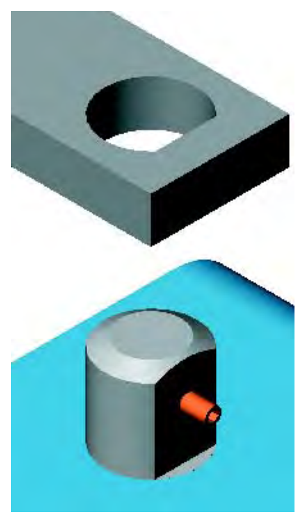
9 Insert a set screw.
Insert the existing component named set screw. Mate it to the small hole in the crank-shaft with a **Concentric** mate.

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.



Lesson 13
Using Assemblies

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

- Perform mass properties calculations.
- Create an exploded view of an assembly.
- Add explode lines.
- Generate a bill of materials for an assembly.
- Copy a bill of materials to a drawing.

Using Assemblies

This lesson will examine other aspects of assembly modeling using a version of the universal joint and other assemblies. 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 assembly 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.
- **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 in the assembly and copied into 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

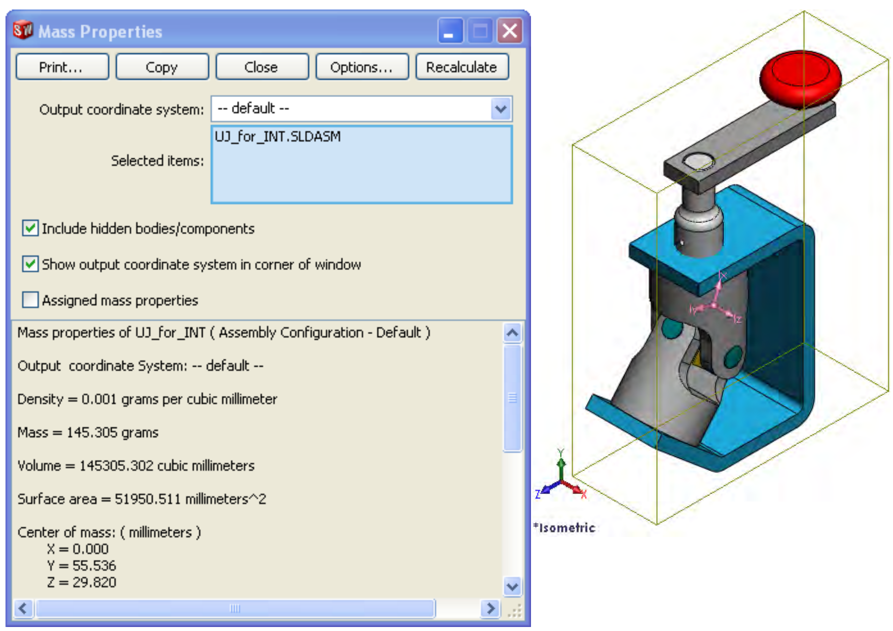
Mass properties calculations were introduced earlier in this course. When working with assemblies, the important thing to remember is that the material properties of each component are controlled individually via the **Material** feature in the part. The material properties can also be set through **Edit Material**.

1 **Open existing assembly.**
Open the existing assembly UJ_for_INT.

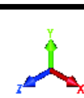
2 **Mass properties.**
Click **Mass Properties** on the Tools toolbar.

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 can be directed to check all components in the assembly, or just selected ones.

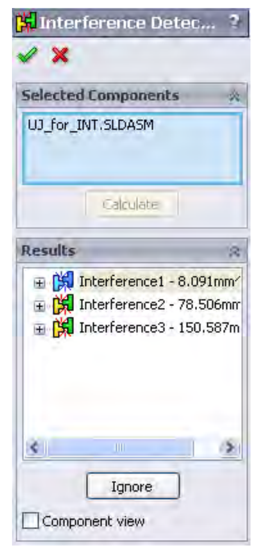
Where to Find It

- Click **Interference Detection** on the Assembly toolbar.
- From the **Tools** menu choose: **Interference Detection...**.

---

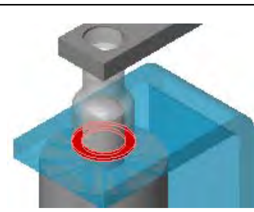
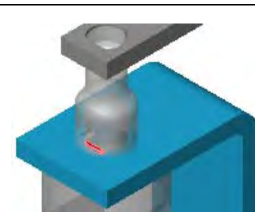
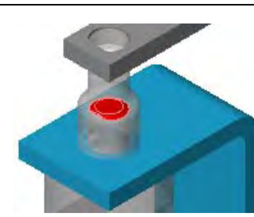
4 Click Tools, Interference Detection....
The **Interference Detection** PropertyManager opens.

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.



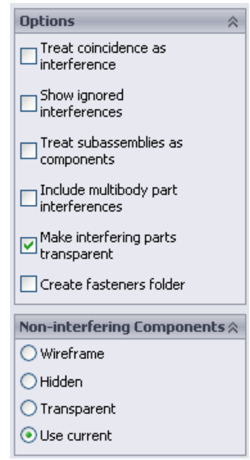
Click **Calculate**.

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**.

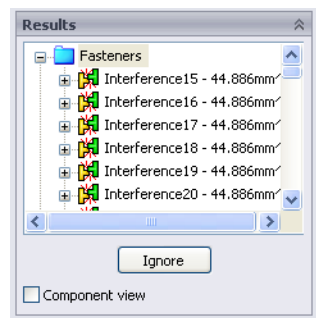
Interference1	Interference2	Interference3
		
bracket	Yoke_male	Yoke_male
crank-shaft	crank-shaft	crank-shaft

Interference Options

The **Options** group box selections are used to refine the detection criteria.



- **Treat coincidence as interference** includes all coincident faces as interferences.
- Selected interferences can be flagged to be ignored by the calculations using the **Ignore** button . They can be shown later using **Show ignored interferences**.
- Clicking **Treat sub-assemblies as components** ignores any interferences within the sub-assembly itself and uses it as a single component.
- **Include multibody part interferences** looks at body to body interferences within part components.
- Using **Make interfering parts transparent** shows the interference volumes in a transparent state.
- **Create fasteners folder** generates a `Fasteners` folder to hold all the interferences that involve a fastener.

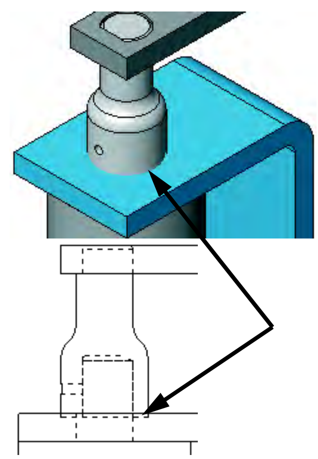


The **Non-interfering Components** group box is used to determine how the components that do not interfere are displayed.

7 Visual methods.

Areas of interference can sometimes be determined visually. **Shaded** (without edges) and **Hidden Lines Visible** displays can be used.

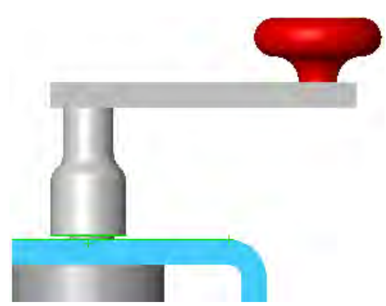
In this case, the `crank-shaft` volume overlaps that of the `bracket`.



8 Flip dimension.

Right-click the `Distance1` mate and choose **Flip Dimension**.

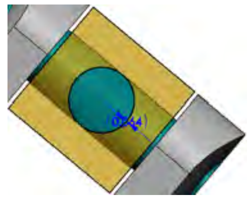
You can also right-click the mate and choose Edit Feature to edit options in the PropertyManager including **Flip Dimension**.

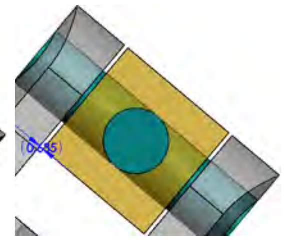


9 Recheck the interferences.
Select the `bracket`, `crank-shaft` and `Yoke_male` components and click **Interference Detection**. As expected, `No Interference` is the result.

Checking for Clearances

The actual clearance between components, like interferences, may be difficult to determine visually. Clearances between both parallel and concentric components can be checked.





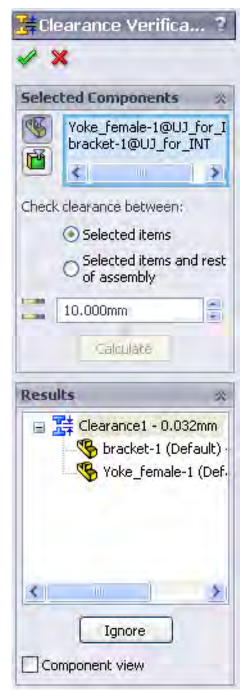
Introducing: Clearance Verification

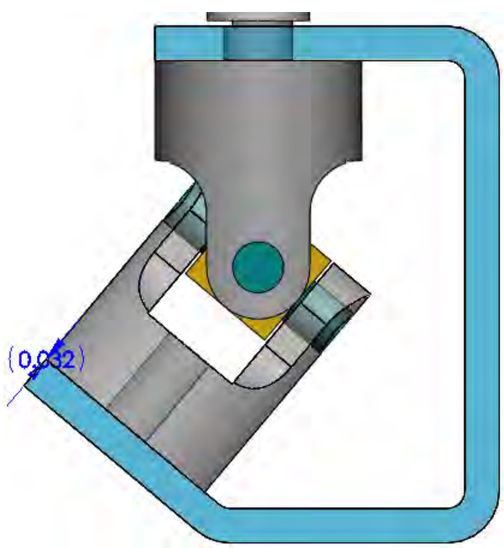
Clearance Verification is used to determine the static clearance between component parts in an assembly. It can be directed to check the clearance between selected components in the assembly, or all components against each other.

Where to Find It

- Click **Clearance Verification** on the Assembly toolbar.
- From the **Tools** menu choose: **Clearance Verification...**.

10 Clearance verification.
Click **Clearance Verification** and select components `Yoke_female` and `bracket`. Set the **Minimum Acceptable Clearance** value to **10mm** (larger than expected). Click **Calculate** and the clearance will appear as `Clearance1` in the **Results**. Click **OK**.





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

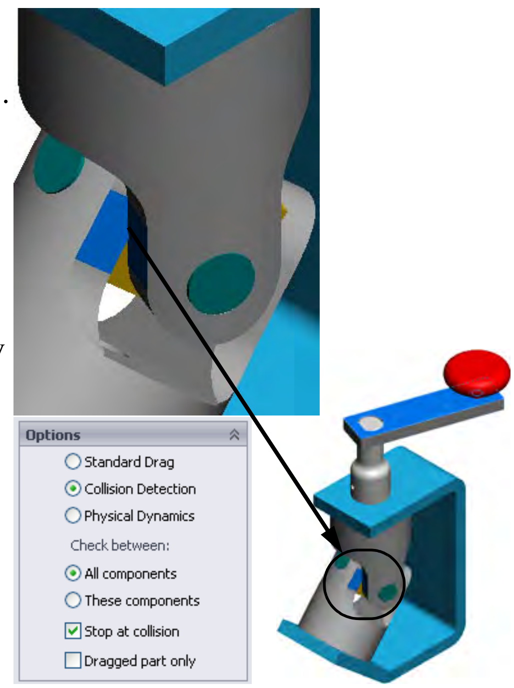
- On the **Move Component** or **Rotate Component** PropertyManagers, select **Collision Detection**.

---

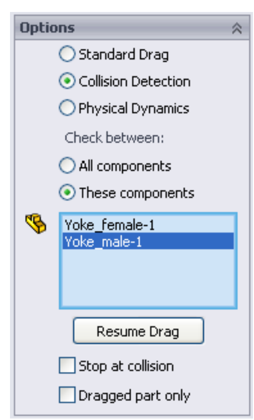
11 Collision Detection.
Click **Move Component** and select **Collision Detection**.

Select **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.



12 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**.

13 Turn off Collision Detection.
Click **OK** to close the PropertyManager.

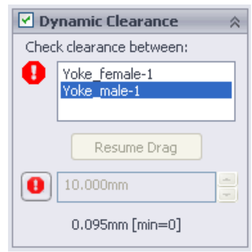
Performance Considerations

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.

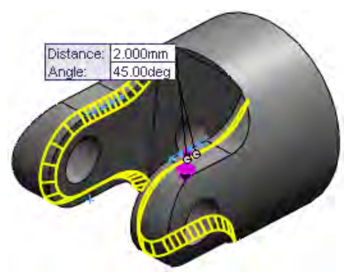


Correcting the Interference

Filleting or chamfering the edges of the yokes will eliminate the interference.

14 Open part.
In the FeatureManager design tree, right-click the Yoke_female and select **Open Part**.

Add a **2mm x 45 chamfer** to the edges as shown. Save the changes.



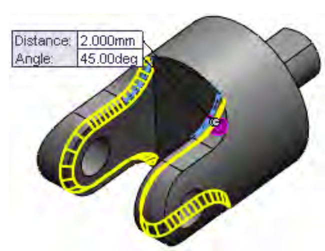
15 Return to the assembly.
Click **Window, UJ_for_INT.SLDASM** or use **Ctrl+Tab**.

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.

16 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.



17 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.

18 Turn off the Move Component tool.

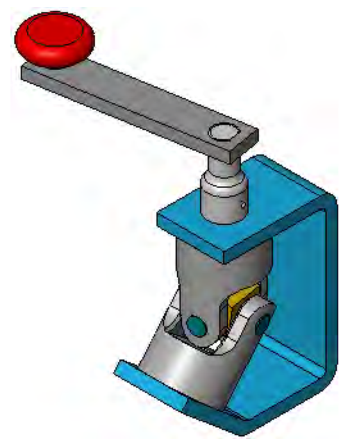
Changing the Values of Dimensions

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.

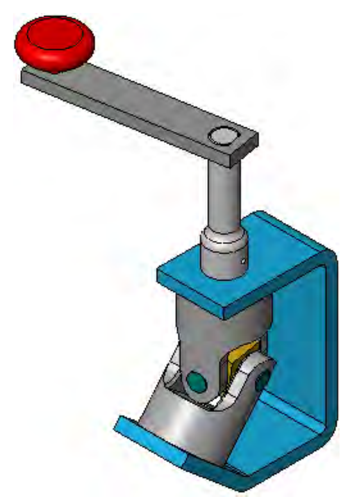
The feature can be double-clicked from the FeatureManager design tree or the screen, but the dimension will always appear on the screen.

19 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 **100mm**.

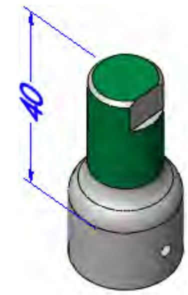


20 Edit the crank-shaft.
Change the value of the length to **65mm**. Notice that not only are the parts rebuilt and the assembly updated, the mating relationships ensure that the crank-arm moves up when the crank-shaft gets taller and the crank-knob moves when the crank-arm gets longer.



21 Open crank-shaft.
Right-click the crank-shaft and select **Open Part** from the shortcut menu.

22 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 **40mm** and close the part, saving the changes.

23 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**.

24 Change values back.
Select and change the dimension of the crank-arm to **75mm** and rebuild.

Exploded Assemblies

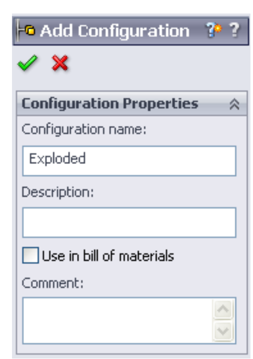
You can make **Exploded Views** of assemblies automatically or by exploding the assembly component by component. The assembly can then be toggled between normal and exploded view states. Once created, the **Exploded View** can be edited and also used within a drawing. **Exploded Views** are saved with the active configuration.

Setup for the Exploded View

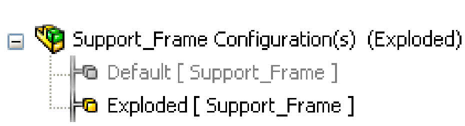
Before adding the **Exploded View**, there are some setup steps that will make the exploded view easier to access. It is good practice to create a configuration for storing an **Exploded View** and also to add a mate that holds the assembly in a "starting position".

1 Open an assembly.
Open the assembly Support_Frame.sldasm located in the Exploded_Views folder.

2 Add a new configuration.
Switch to the ConfigurationManager, right-click and select **Add Configuration**.



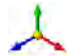
Type the name Exploded and add the configuration.



The new configuration is the active one.

For more information on *Assembly Configurations*, see the *Assembly Modeling* training 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

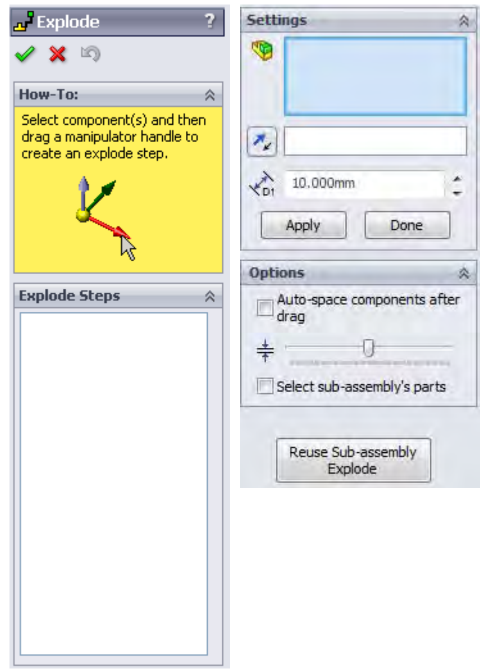
- From the **Insert** menu, pick **Exploded View...**.
- Or, click **Exploded View**  on the Assembly toolbar.

---

3 Click Insert, Exploded View.
The **Exploded View** PropertyManager 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.

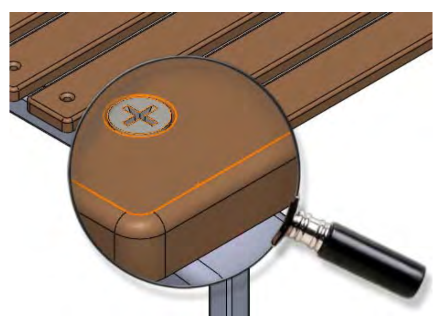
The **Options** group box includes the automation Auto-space and sub-assembly options.

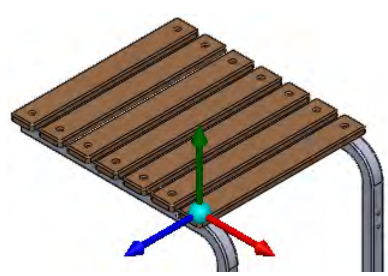


Exploding a Single Component

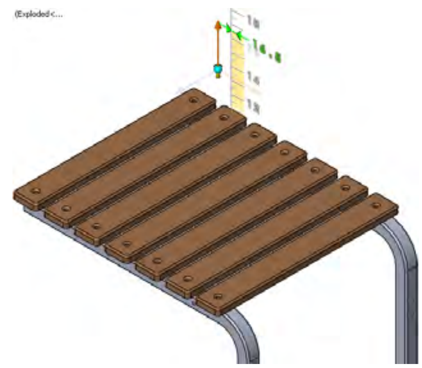
One or more components can be moved in one or more directions. Each movement (one or more components) set by a distance and direction is considered one step.

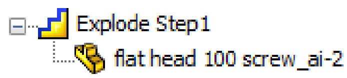
4 Select component.
Use the magnifying glass to select the `Flat Head Screw` component. A Move Manipulator appears at the selection. The Move Manipulator is aligned with the axes of the component.





5 Drag explode.
Explode the component by dragging the green leg away from the assembly and using the ruler to set the distance. The `Explode Step1` feature is added. The components are listed beneath it. Click off the component to complete the step.

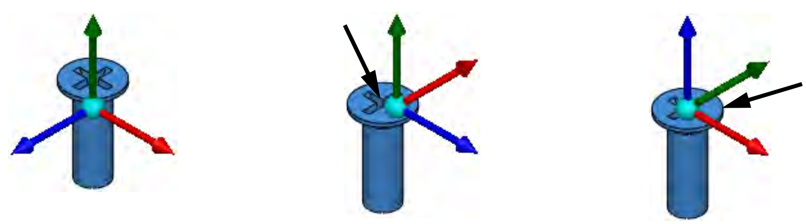




Tip Selecting the step by name in the dialog displays the components in magenta with the blue arrow.

Move Manipulator

If the **Move Manipulator** axes do not point in the desired directions, its orientation can be changed. Right-click the manipulator origin and choose **Move to selection**. Select an edge, axis, face or plane to reorient it.

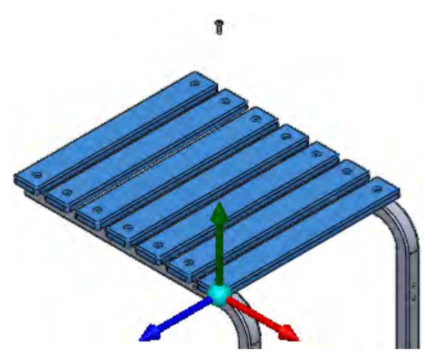


Multiple Component Explode

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.

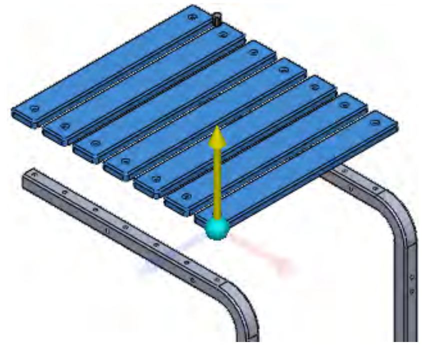
---

6 Selection.
Select all of the `side_table_plank_wood` components as shown. The last selection produces the move manipulator as shown.



Tip Making a multiple component selection can be made by clicking each one or using a drag-select window.

7 Move multiple.
Move the components along the green leg as shown.



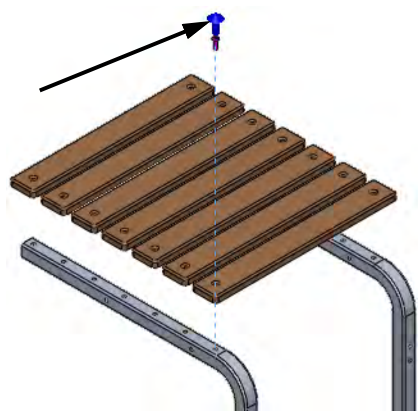
Drag Arrow

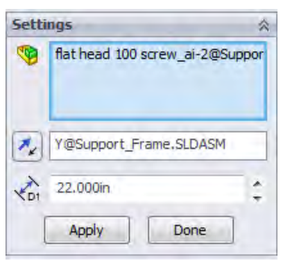
The **Drag Arrow** is used as a vector for the explode step. Once created, the step distance can be modified by clicking the step in the **Explode Steps** dialog and dragging the blue arrow along the explode line.

Edit Step

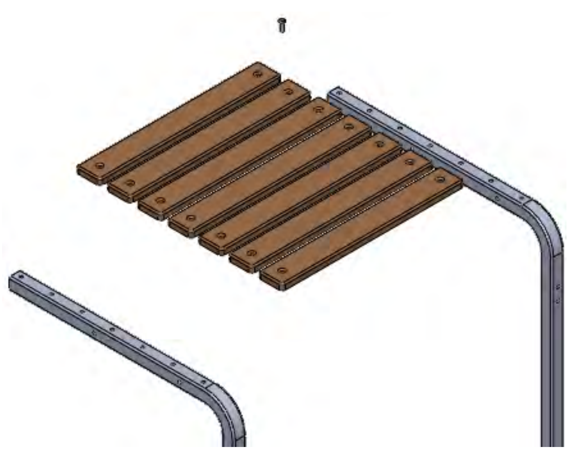
Right-click a step and choose **Edit Step** to edit which components are used in the step or to set the distance to an exact numeric value.

8 Adjust step.
Click `Explode Step1` in the **Explode Steps** dialog. Drag the blue arrow upwards to adjust the explode distance. Right-click the same step and choose **Edit Step**. Set the distance to 22in, then click **Apply** and **Done**.

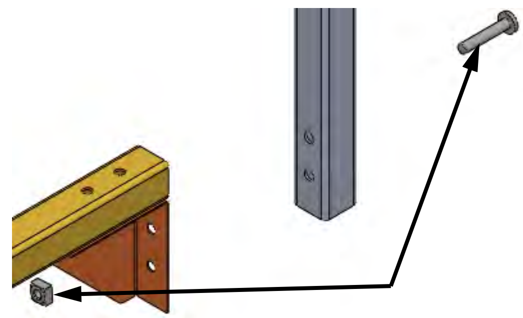




9 New steps.
Select the `Support_Leg<1>` and explode it away from the center as shown. Do the same for the `Support_Leg<2>` in the opposite direction.



10 Hardware.
Explode the `binding head screw_ai` <3> and `square nut_61_ai` <3> components as shown using separate steps.



Sub-assembly Component Explode

Sub-assemblies can be treated in several ways. By default they are treated as single components and are moved as one. By clicking **Select sub-assembly's parts**, they are treated as individual components, and each can be moved independently.

Auto-spacing

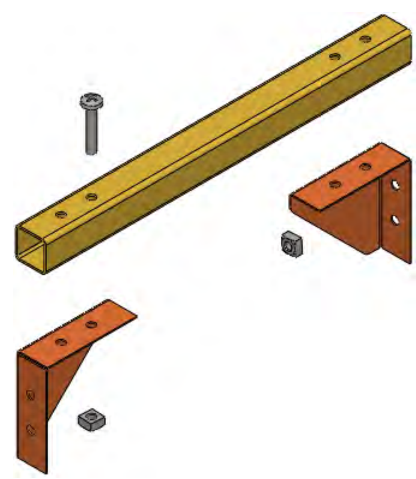
The **Auto-space components after drag** option is used to spread a series of components along a single axial step. The spacing can be set with a slider and changed after creation.

Note

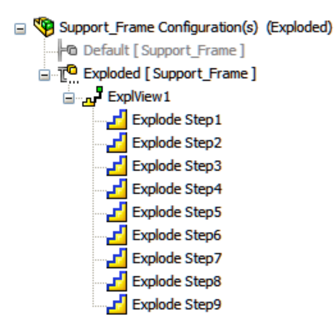
If an exploded view of a sub-assembly already exists, it can be added to the current exploded view by using **Reuse Sub-assembly Explode**.

11 Sub-assembly components.
Click **Select sub-assembly's parts**. Select the `binding head screw_ai` <1> component and create a step as shown.

Repeat the procedure to create steps for the `square nut_61_ai` <1> and `Brace_Cross_Bar` <1> components.



12 Complete explode.
Click **OK** to complete the explode. Expand the `ExplView1` folder to display the explode steps.



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.

Explode Line Selections

Typical selections, such as vertices, edges and faces, can be made to create explode lines. It is important to:

- Select geometry in the order to define the explode line.
- Select geometry that is appropriate to start, end or pass through geometry.

Vertices and edges are typically suited start and end explode lines. Faces are typically used to "pass through". These selections will be shown in the following steps.

Introducing: Explode Line Sketch

An **Explode Line Sketch** enables 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

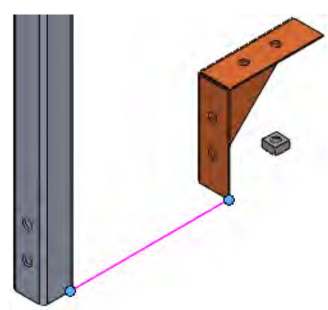
- On the **Insert** menu, click **Explode Line Sketch**.
- Or, click **Explode Line Sketch** on the Assembly toolbar.

---

13 Route line.

Click **Explode Line Sketch** to start the 3D sketch. Select the vertices as shown to create a route line between them. Various combinations of the **Options** can be used to get different results.

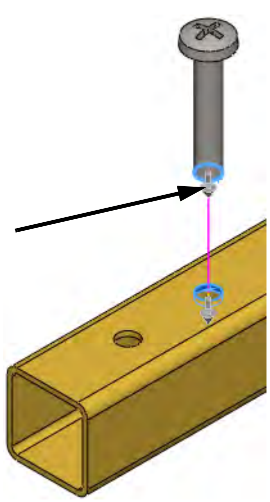
Click **OK**.



14 Start explode line.

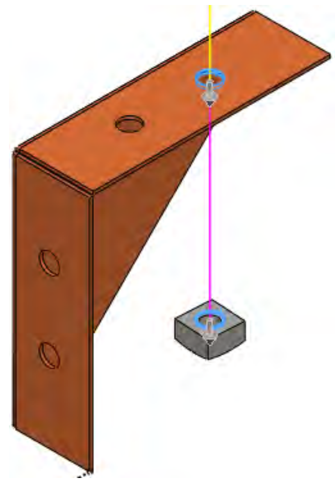
Select (in order) the circular edge of the `binding head screw_ai <1>` to start the line. Next, select the cylindrical face of the hole in the `Brace_Cross_Bar <1>` component to pass through.

Click the grey arrow at the first selection to set the direction.

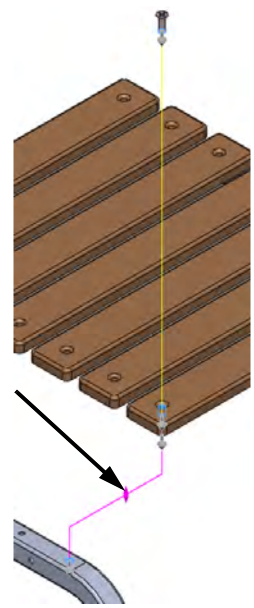


15 End explode line.
Select the cylindrical face of the hole in the `Brace_Corner <1>` component to pass through. To end the line, select the circular edge of the `square nut_61_ai <1>`.

Click **OK**.

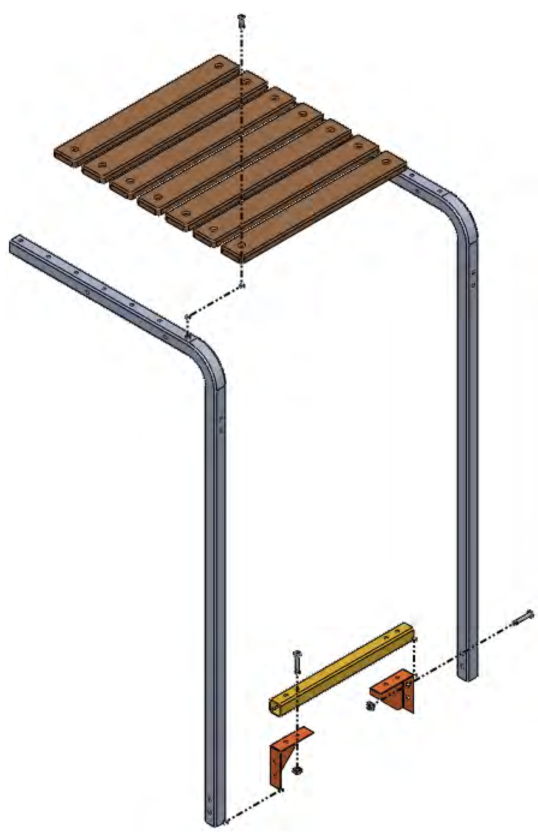


16 Adjusting lines.
Add another explode line as shown. The bend lines that are added can be modified by selecting and dragging. Double arrowheads appear on movable lines when they are selected.



17 Save.
Click **OK** to complete the process of adding explode lines. Save the changes.

More lines can be added if desired.



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 `ExplView1` feature in the ConfigurationManager.
- If the exploded view is collapsed, right-click **Animate Explode** from the `ExplView1` feature.

Animation Controller

The Animation Controller is invoked by the **Play** 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

18 Animation toolbar.
Right-click on `ExplView1` and choose **Animate Collapse**. The dialog uses standard VCR-like controls including **Play**.

19 Save.
Save the assembly after collapsing the exploded view. Do not close the assembly.

Bill of Materials

In an assembly, a bill of materials report can be automatically created and edited. It can then be inserted onto the drawing sheet. The finished version of the BOM will appear in the assembly and later on the drawing sheet.

Where to Find It

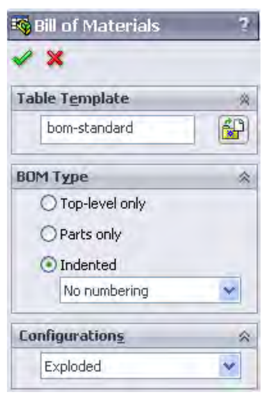
- Click **Bill of Materials** on the Table toolbar.
- Or from the **Insert** menu, select **Tables, Bill of Materials...**.

---

20 BOM Settings.
Click **Insert, Tables, Bill of Materials...**.

Select bom-standard as the **Table Template** and **Indented** as the **BOM Type**.

Click **OK** then click in the graphics window to place the BOM.

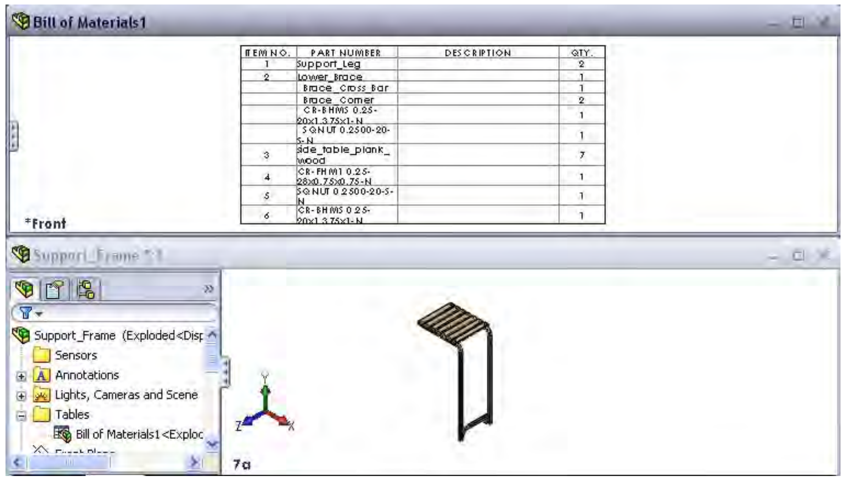


21 BOM feature.
Expand the Tables folder. The bill of materials feature, Bill of Materials1 <Exploded>, is stored there.

The Exploded text in braces refer to the configuration of the assembly.

22 Show table in new window.

Right-click the table `Bill of Materials1 <Exploded>` and choose **Show table in new window**. Click **Window, Tile Horizontally** to show both windows.



23 Moving a column.

Click in the `QTY.` column and click the header cell **D**. Drag the column header cell to the left, dropping it at the position shown. Click in the table and drag vertical or horizontal lines to resize the cells.

A	B	C	D
ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Support_Leg		2
2	Lower_Brace		1
	Brace_Cross_Bar		1
	Brace_Corner		2
	CR-BHMS 0.25-20x1.375x1-N		1
	SQNUT 0.2500-20-S-N		1
3	side_table_plank_wood		7
4	CR-FHM1 0.25-28x0.75x0.75-N		1
5	SQNUT 0.2500-20-S-N		1
6	CR-BHMS 0.25-20x1.375x1-N		1

A	B	C	D
ITEM NO.	QTY.	PART NUMBER	DESCRIPTION
1	2	Support_Leg	
2	1	Lower_Brace	
	1	Brace_Cross_Bar	
	2	Brace_Corner	
	1	CR-BHMS 0.25-20x1.375x1-N	
	1	SQNUT 0.2500-20-S-N	
3	7	side_table_plank_wood	
4	1	CR-FHM1 0.25-28x0.75x0.75-N	
5	1	SQNUT 0.2500-20-S-N	
6	1	CR-BHMS 0.25-20x1.375x1-N	

Assembly Drawings

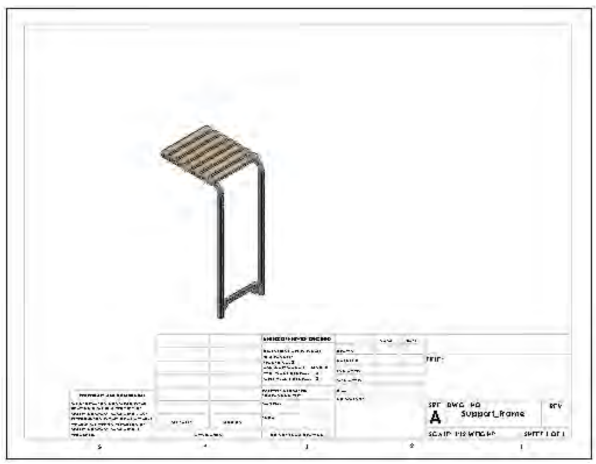
Assemblies have several unique requirements when it comes to making detail drawings of them. In addition to specialized views, assemblies require a Bill of Material and Balloons to fully document the assembly. In this example, the Bill of Material table created in the assembly will be copied to the drawing.

1 **New drawing.**

Use **Make Drawing from Part/Assembly** to create a new drawing using the `A-Scale1to2` template.

2 **Model View.**

Drag and drop the `*Isometric` view from the **View Palette**. Set the **Display Style** to **Shaded with Edges**.



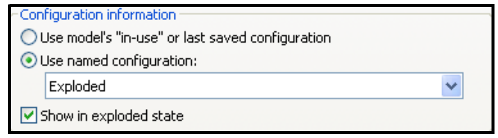
Displaying Exploded Views

Views are normally created in their non-exploded state. To display their exploded state, the configuration that contains the exploded view must be selected with the **Show in exploded state** option.

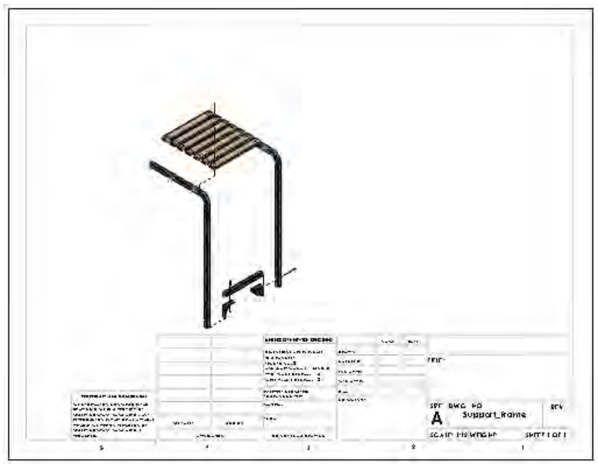
Note

The **Show in exploded state** option will be available *only* if there is an existing exploded view in the selected configuration.

3 **Exploded state.**
Right-click the view and click **Properties**. Click **Use named configuration** and select the `Exploded` configuration.



Click **Show in exploded state** and click **OK**.

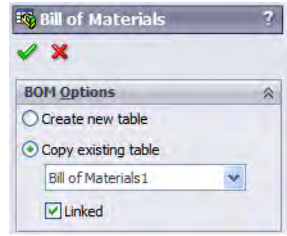


Copying a BOM Table from the Assembly

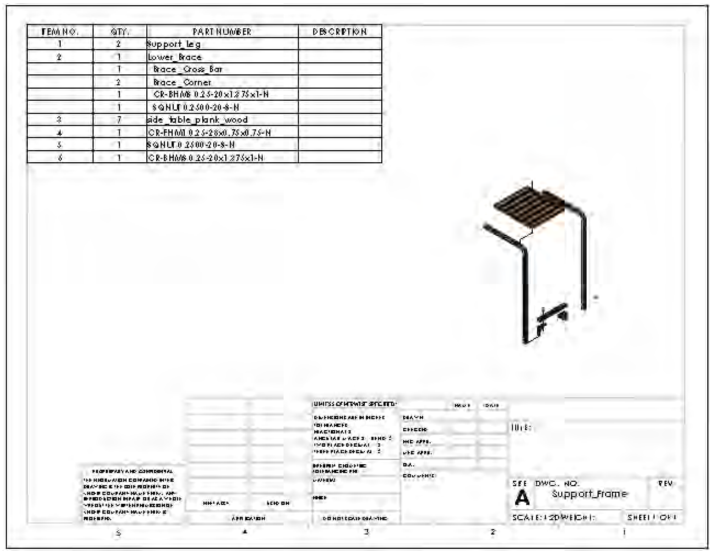
If a bill of materials table has been created in the assembly, it can be copied to the drawing.

The same option used to create the table is used to copy it. For more information, see *Bill of Materials* on page 468.

4 Copy table.
Click **Insert, Tables, Bill of Materials** and click in the view. Click **Copy existing table** `Bill of Materials1`, click **Linked** and click **OK**.



Move the Bill of Materials to the top left corner of the drawing format and click to place it.



Note For more information about creating and editing Bill of Material tables, see the *SolidWorks Drawings* manual.

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: Balloons

The **Balloon** command is used to label the components of an assembly drawing by item number and optionally quantity. There are several different shapes of balloons.

Where to Find It

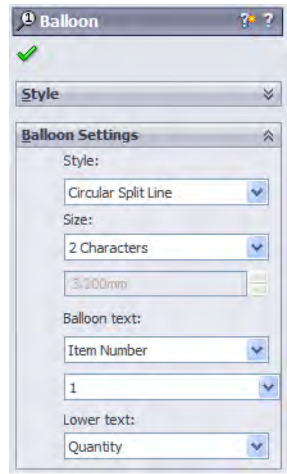
- On the Annotations toolbar, click **Balloon**.
- Or, on the **Insert** menu, click **Annotations, Balloon...**.

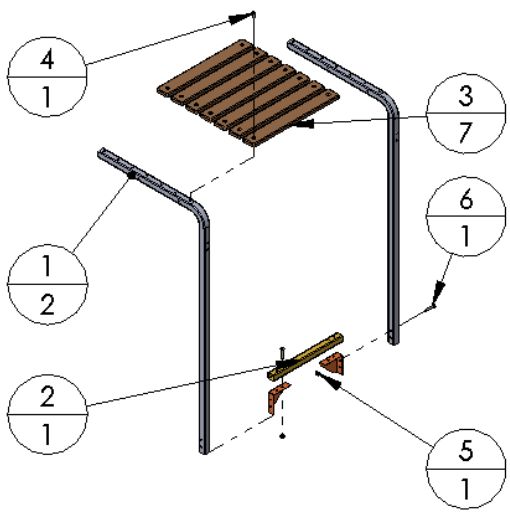
---

5 Insert balloons.

Click on the **Balloon** tool and select the **Circular Split Line** style with **Item Number** and **Quantity**.

Click on an edge, vertex or face of the geometry then click to place the balloon.

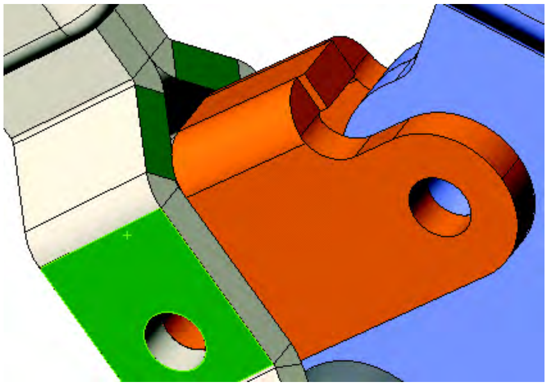




6 Save and close the drawing and any other open files.

Exercise 60: Using Collision Detection

Using the assembly provided, determine the range of motion of the clamp handle.



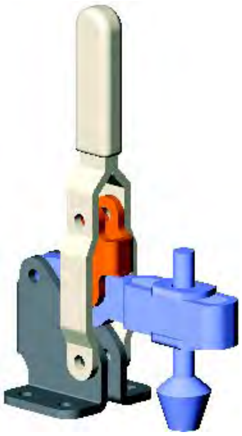
This lab reinforces the following skills:

- *Checking for Interference* on page 454.
- *Introducing: Collision Detection* on page 457.

Procedure

Open an existing assembly.

1 Existing assembly.
Open the existing assembly named Collision from the folder Collision.



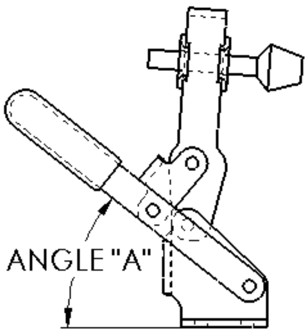
2 Collision locations.
The link stops the motion of the assembly in two places. Move the assembly to the point of collision and measure the angle formed using **Measure** or dimensions on a drawing view.

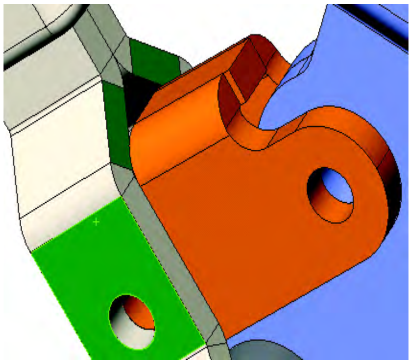
ANGLE "A"- As the handle sub-assy is pulled back, the link hits it.

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 61: Checking for Interferences, Collisions and Clearances

Using the assembly provided, check for interferences, clearances and collisions.

This lab reinforces the following skills:

- *Checking for Interference* on page 454.
- *Checking for Clearances* on page 456.
- *Introducing: Collision Detection* on page 457.

Procedure

Open an existing assembly.

1. **Existing assembly.**
Open the existing assembly named A_D_Support from the folder Clearances.

2. **Static interference.**
Check for static interferences using **Interference Detection**.

3. **Dynamic clearance.**
Drag the Internal sub-assembly. Collisions stop the motion in two places. Move the assembly to the point of collision (open) and measure the minimum distance between the End and the small collar using the **Dynamic Clearance** option.

(225mm)

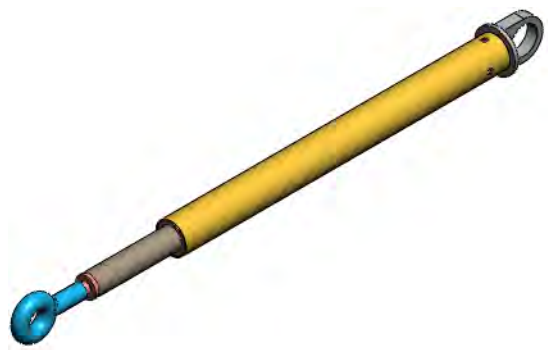
4. **Sub-assembly clearances.**
Open the sub-assembly internal and make sure that there are clearances between:

- Components small center_tube and small collar.
- Components small center_tube and thin_collar.

(0.13mm and 0.14mm)

5 Top level assembly clearances.
Return to the top level assembly A_D_Support and make sure that there are clearances between:

- Components center_tube and small center_tube.
- Components center_tube and small collar.



(3.68mm and 0.10mm)

6 Save and close all files.

Exercise 62: Exploded Views and Assembly Drawings

Using the existing assemblies, add exploded views, explode lines and a BOM in the assembly. Use the exploded views to generate drawings with balloons and copy the BOM from the assembly. Use the `A-Scale1to2` template.

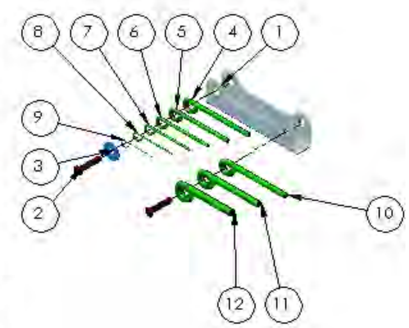
This lab reinforces the following skills:

- *Exploded Assemblies* on page 460.
- *Explode Line Sketch* on page 465.
- *Assembly Drawings* on page 470.
- *Bill of Materials* on page 468.

The files are found in the `Exploded Views` folder.

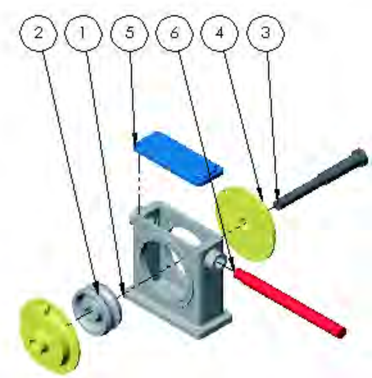
Assembly: **part configs**

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



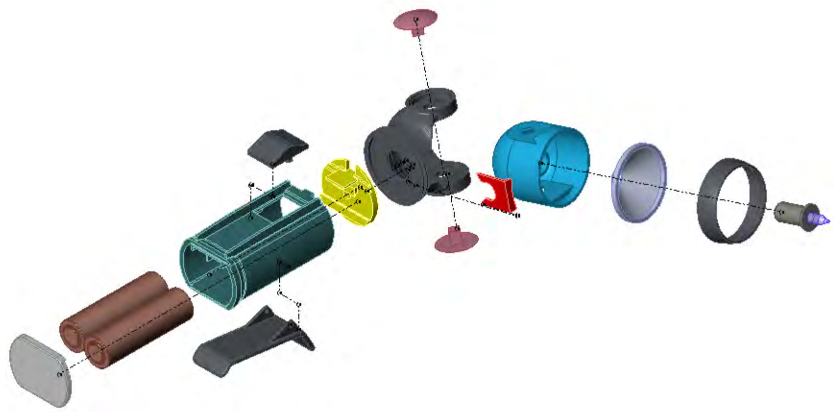
Assembly: **Gearbox Assembly**

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



Exercise 63: Exploded Views

Using the existing assemblies, add exploded views and explode lines.



This lab reinforces the following skills:

- *Exploded Assemblies* on page 460.
- *Explode Line Sketch* on page 465.

Procedure

Open an existing assembly.

1 Existing assembly.
Open the existing assembly named `Flashlight` from the folder `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.

Your explode line sketch may vary significantly based on explode distances and selected entities.

2 Create Exploded View.
Create exploded view and explode line sketch.

Appendix A
Templates

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 saved 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**. These are tabs within **Options** to make changes to the system or document properties. This enables 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 any 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.

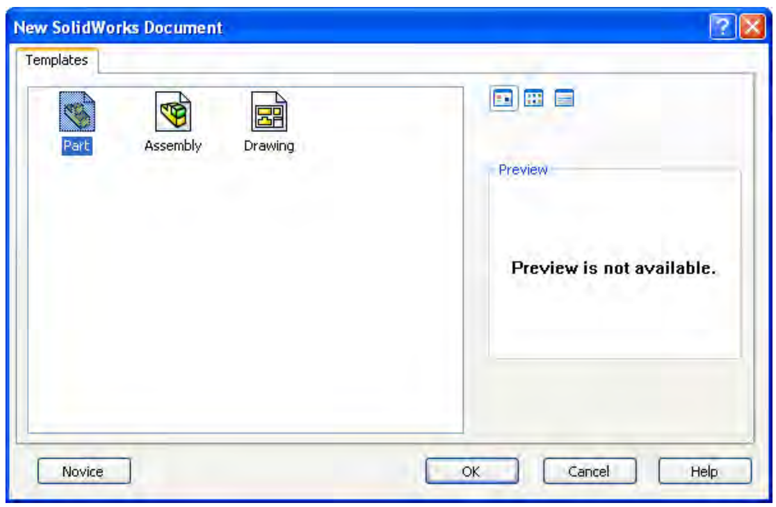
How to Create a Part Template

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 a 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**:

- **Drafting Standard**
  Overall drafting standard: ANSI.
- **Dimensions, Font**
  Dimension: Century Gothic; Height = 12 points.
- **Annotations, Font**
  Balloons: Century Gothic; Height = 12 points.

  Notes: Century Gothic; Height = 12 points.
- **Dimensions, Primary precision**
  Primary dimension, Value: 3.

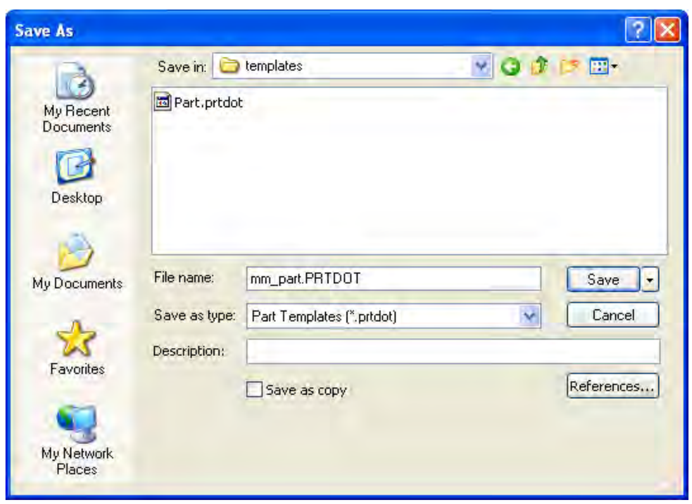
- **Grid/Snap**
  Display Grid - Disabled.
- **Units**
  Unit system - MMGS.
- **Reference Geometry**
  The default names for the three system planes are not controlled by **Tools, Options**. They are controlled by the document template. Since most feature can be renamed, the planes can be renamed as 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 planes. For example, you might prefer XY, XZ, and YZ instead of the default names.

4 Save a template.
Click **File, Save As...**.

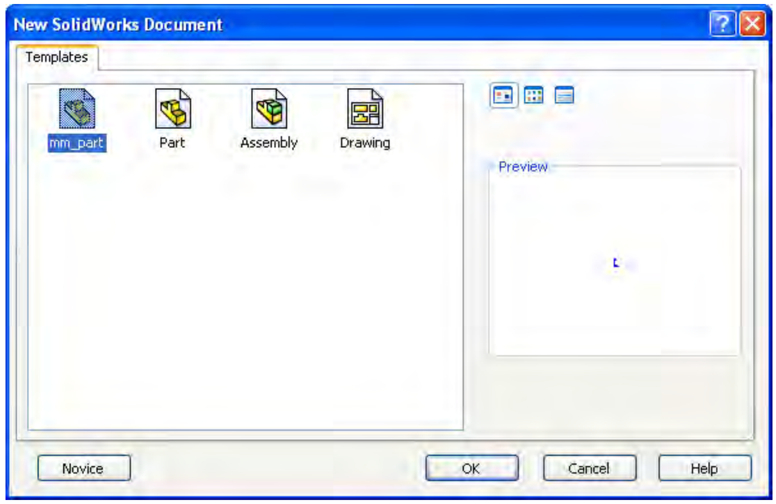
For **Save as type**, select **Part Templates**.

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 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.

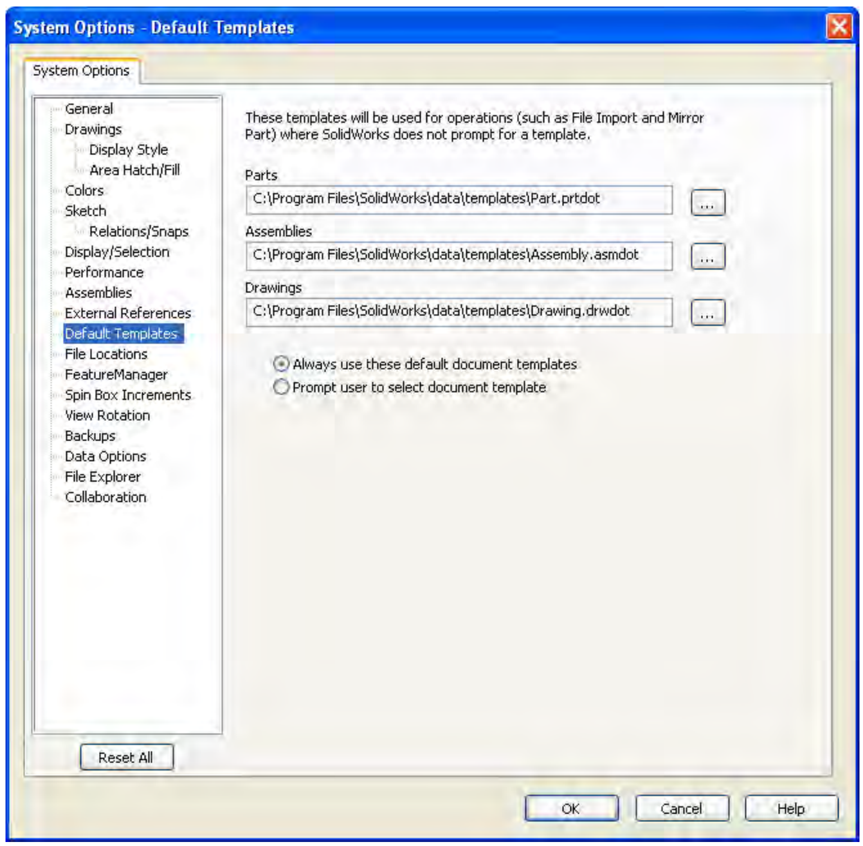
You can control where SolidWorks searches for the templates by means of **Tools, Options, System Options, File Locations**. The **Show folders for** box displays search paths for files of various types, including document templates. The folders are searched in the order they are listed. You can add new 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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