

INTRODUCTION TO SOLIDWORKS (HORTMAN)



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Introduction to SolidWorks (Hortman)

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Licensing

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CHAPTER OVERVIEW

1: SolidWorks User Interface

Learning Objectives

By the end of this chapter you will be able to...

- Open the SolidWorks software.
- Identify the critical elements of the user interface.
- Change SolidWorks settings
- Manipulate the SolidWorks graphics area and edit models.

This chapter will introduce you to what SolidWorks is, what it is used for, and how the software works. You will learn how to start a new part file, the different file types, the user interface, different setting and view options.

[1.1: What is SolidWorks?](#)

[1.2: General User Interface Components](#)

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1.1: What is SolidWorks?

SolidWorks is a 3D parametric design software that is used to design all sorts of products such as automobiles, marine equipment, airplane parts, cell phones, cameras, furniture, electrical assemblies, glasses, lighting fixtures, toys, vacuum cleaners, or any other like product you can think of. SolidWorks is typically used to design assemblies that work mechanically that are typically less than 200 parts.

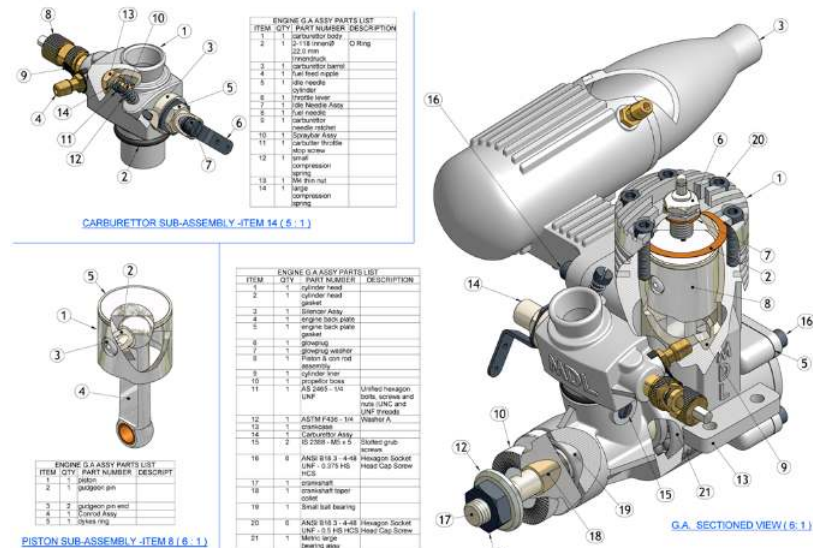


Figure 1.1.0: Small Engine

What is parametric design? It is a constraints-based modeling system where geometry is drawn, then “anchored” with dimensions and relations to clearly communicate design intent. This is very different from typical 2D CAD systems where geometry is drawn to the desired constraints, but nothing is constraining the geometry to the form, orientation, location, and size desired. In a parametric modeler, the size, form, orientation, and location are specified.

For example, a horizontal 2-inch line drawn in a 2D design software is just a line that can be moved and resized and reoriented by the user (Figure 1.1.1). But a horizontal 2-inch line in SolidWorks is constrained to be horizontal and constrained to be 2 inches (Figure 1.1.2).



Figure 1.1.1: 2D CAD line drawn horizontally to 2 inches.

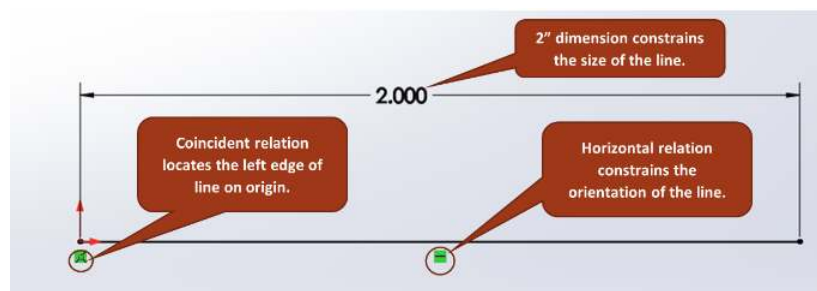


Figure 1.1.2: Line drawn in SolidWorks horizontally to 2 inches.

SOLIDWORKS SYSTEM REQUIREMENTS

SolidWorks has strict system requirements, meaning that you need a specific type of computer with the right components to be able to download and use SolidWorks. The SolidWorks system requirements and Graphics Card requirements are available on the SolidWorks website and are linked below:

- [SolidWorks System Requirements](#)
- [SolidWorks Video Card Requirements](#)

SOLIDWORKS FILES: PARTS, ASSEMBLIES, AND DRAWINGS

There are 3 primary SolidWorks file types (see figure 1.1.3: and 1.1.4:):

- Parts
- Assemblies
- Drawings

Mechanical Designs are typically composed of one or more assemblies. Assemblies are made from parts and the design of parts are typically conveyed through 2D technical drawings.

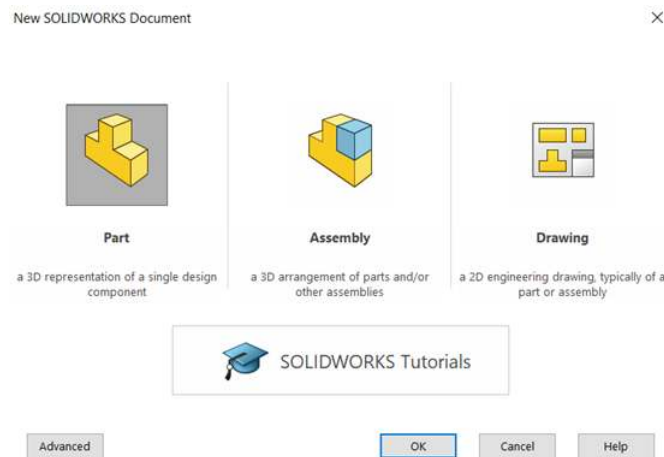


Figure 1.1.3: SolidWorks new dialog box showing part, assembly, and drawing file types.

This makes SolidWorks parts the building blocks of all other SolidWorks file types. Parts are created and then used to create drawings and assemblies (and subassemblies in larger products) to complete product definition and to maintain a link between the individual parts, drawings, and assemblies. This allows for drawings and assemblies to be updated whenever part files are updated, eliminating redundant design revisions.

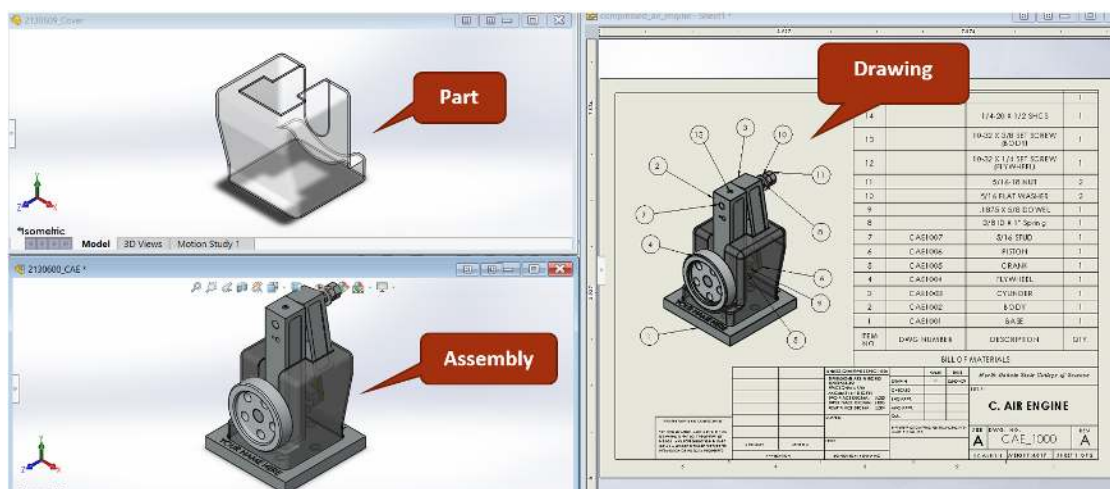


Figure 1.1.4: Part, Assembly, and Drawing shown in SolidWorks

Other file types can be opened in SolidWorks as well such as STEP files, 3D printing files, or other file types. This is so that models created or modified in other platforms can be shared and collaborated on with stakeholders having different modeling software. STEP and other neutral format file types are especially helpful for protecting proprietary information when sharing designs with suppliers and other stakeholders.

FILE MANAGEMENT WITH SOLIDWORKS FILES

Because SolidWorks files are interlinked (such as assemblies and drawings being linked to part files such that a change in one will update the rest), it is crucial that files be moved, copied, and renamed with care. The most common methods for doing each is as follows:

- *Rename*: Rename within the SolidWorks software instead of via windows file explorer.
- *Move/Copy*: Pack and Go the assembly/drawing to the new location desired.
- *Major updates to dozens of files*: Use SolidWorks® Explorer to manage files (discussed later in part 1).

Do not rename files using Windows® File Explorer and do not move files without using Pack and Go!

? Exercise 1.1.1: ? Working with SolidWorks Files

Now let's try and work with SolidWorks files ourselves. Download and extract the [Coupler part and drawing files](#) and change the name of the part file by following the instructions in the video below for changing file names.

1. Follow the link to the files.
2. Hit the download button in the top right corner and open the file location.
3. Extract all files.
4. Right click on the part file (SolidWorks Part File Type) and go to the SOLIDWORKS drop down.
5. Select "Rename" and rename the file to whatever you would like.
6. Click OK to confirm or cancel to discard your changes.



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1.2: General User Interface Components

Parts, assemblies, and drawings are created by interacting with the commands in the user interface of SolidWorks. The user interface is composed of the menu bar, CommandManager, graphics area, manager pane, status bar, and task pane (See Figure 1.2.1).

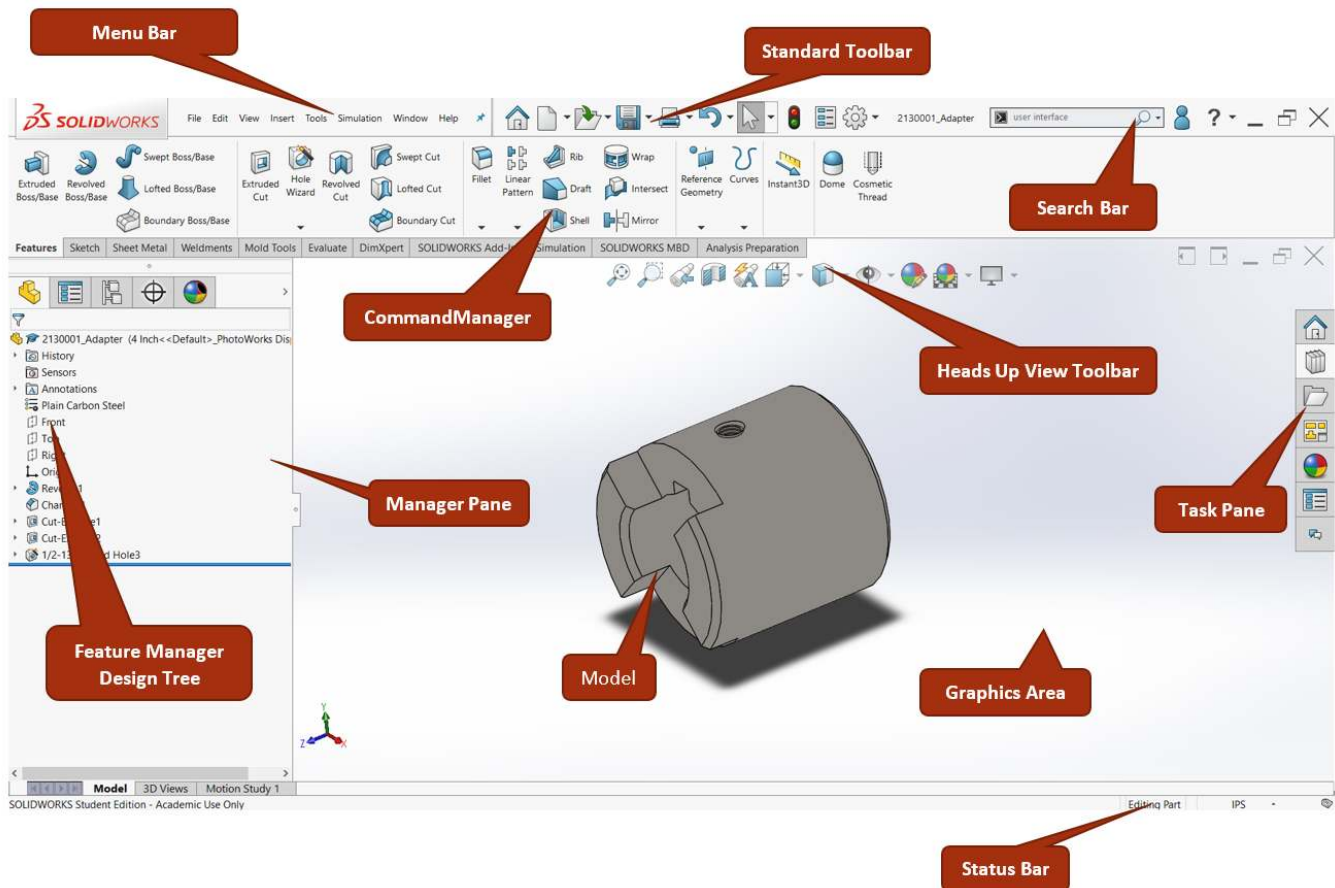


Figure 1.2.1: SolidWorks User Interface

Many of the features in SolidWorks® work like other Windows® programs you are familiar with such as Microsoft Office®, AutoCAD®. For example, if you hover over a button or item in the user interface, a pop-up message will tell you what it is you are hovering over.

More information can be found from [SolidWorks Help user interface content](#). SolidWorks Help library will be very useful for you learning the SolidWorks software because it has answers to many questions on how to do things in SolidWorks. As we learn new terminology related to the SolidWorks software, this textbook will link to a place in the SolidWorks Help library where you can read more information than what this textbook provides.

? Exercise 1.2.1: ? : SolidWorks User Interface

Download and open the [Coupler part file](#) in SolidWorks and interact with it and the SolidWorks User Interface as is shown in the video below.



MENU BAR

The [menu bar](#) is located at the top left of the SolidWorks® user interface and looks very similar to the menu bar in other programs (Figure 1.2.2). All the SolidWorks® commands can be accessed through the menu bar. So, whenever you cannot find a command through any other means, you can find it via the menu bar.



Figure 1.2.2: SolidWorks Menu Bar

The menu bar has 7-8 tabs: **File, Edit, View, Insert, Tools, and Help** which is common to nearly all Windows® programs as well as custom tabs such as **Windows and Simulation**. When you hover over a tab with your mouse cursor, the tab expands. There are other expandable flyouts within each tab as well. Looking at the File tab may be familiar because the options in the file tab are very similar to other programs (Figure 1.2.3).

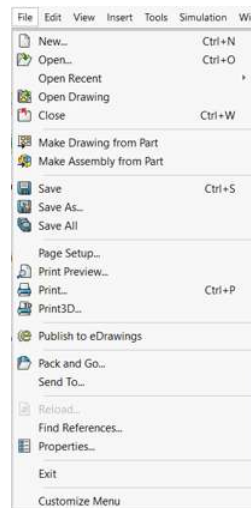


Figure 1.2.3: File Tab of the SolidWorks Menu Bar

STANDARD TOOLBAR

A part of the menu bar is also the **Standard Tool Bar** which has the most used commands related to file management of your SolidWorks® files (Figure 1.2.4).

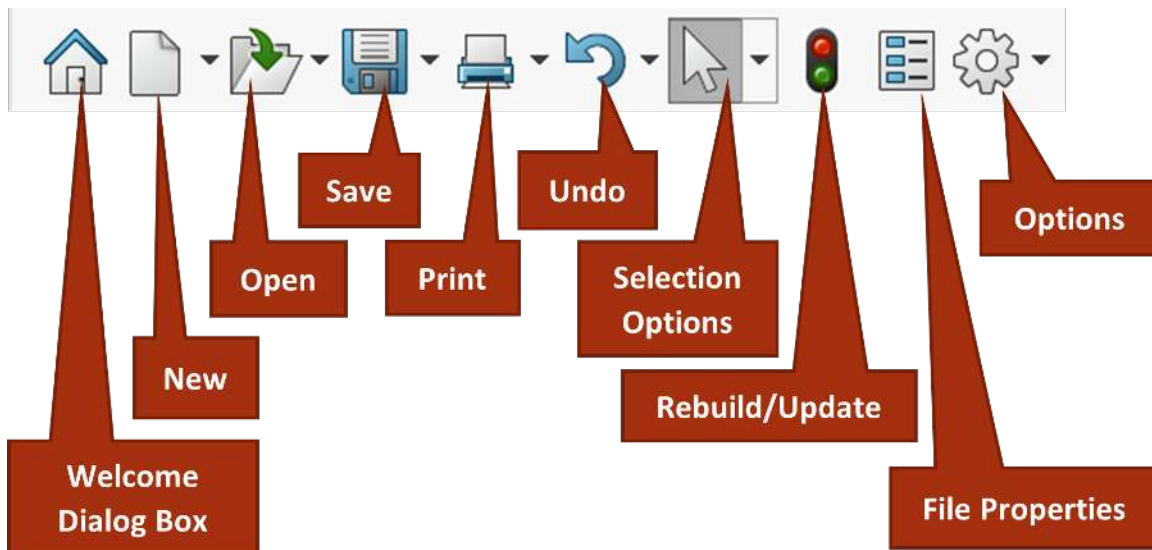


Figure 1.2.4: Standard Tool Bar

The **new**, **open**, **save**, **print**, **undo** buttons work similar to other programs: starting new parts/assemblies/ drawings, opening existing SolidWorks® files, saving your work, printing screenshots of your model or 3D printing your model, undoing the last action in your modeling process.

SELECTION OPTIONS

The [selection options](#) allows you to filter what you are able to select in the model such as only edges, faces, vertex's (points) and also let's you choose between box window selection and lasso window selection (figure 1.2.5). The selection options have other specific selection capabilities which you can discover as you grow in your modeling skills.

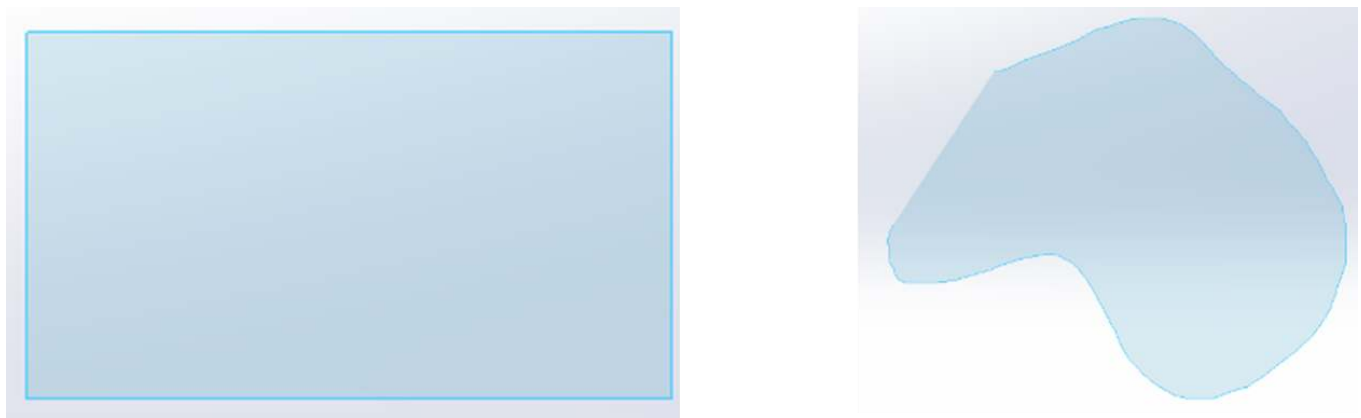


Figure 1.2.5: Rectangular window selection (left) and lasso window selection (right)

REBUILD

The [rebuild](#) button updates the model to reflect most recent model changes. This will be a very often used button in your modeling because you will often make changes to a model and not see those changes reflected immediately. This is due to computer technology limitations. So, to see your model with the most recent updates, you will sometimes need to “rebuild” your model after making a change.

FILE PROPERTIES

The [file properties](#) button opens the file properties dialog box where you can insert *metadata* for your model such as the author of the model, the title, and custom properties such as material designation and weight.

What is metadata in engineering design?

Metadata is the data that is used to describe our 3D model: it is who made the model, the description of the model, the material of the model, the weight, part number, cost; it is pretty much all of the nongeometric data that we want to be attached to the geometric 3D model data. We insert this data using the file properties dialog box so that it can be attached to the model and used downstream in the product lifecycle for raw material ordering, cost projecting, and 2D drawing creation.

OPTIONS

The [Options](#) button opens the options dialog box where you can customize your SolidWorks user experience and set up your SolidWorks environment exactly how you want it. If you ever want a setting to be different, you probably change it via the Options dialog box. From the options drop down, you can also customize the Command Manager, add in SolidWorks® add ins – like Simulation or CAM – and change the size of your command buttons in the SolidWorks® user interface.

SEARCH BAR

Another part of the menu bar is the **Search Bar** which includes a place to search up commands, SolidWorks Forum solutions, files and locations as well as a link to local SolidWorks® help files (Figure 1.2.6).

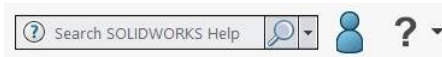


Figure 1.2.6: Search Bar

One particularly helpful part about the search bar is the Command Line drop down option when you can't remember where a particular command is (Figure 1.2.7). The eye to the right of suggested commands in the command bar mode will also show you how to access the command via the menu bar or command manager.

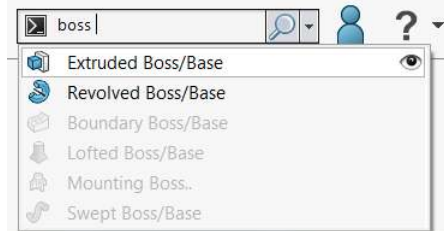


Figure 1.2.7: Using the Search Bar with the Command Line Drop Down Option

? Exercise 1.2.2: ? Exploring the Interface

Get familiar with various components of the SolidWorks User Interface just by trying things out! Some instructions and a tutorial video are below to help you get moving.

1. Try hovering over the different tabs of the menu bar with your coupler part file open and seeing all of the different options in each tab of the menu bar.
2. Try out each of the buttons in the standard toolbar: New, Open, Save, Selection, Print, Options, Rebuild, File Properties and explore what is within each and what you can input into each.
3. Pay particular attention to the Options dialog box when you open options and all of the properties you can enter in the File properties dialog box.
4. Try out the SolidWorks search bar with the general search and the command line drop down.



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1.3: Major User Interface Components

The user interface elements that will be talked about in this section are going to be those that have major impact on your modeling capability in the SolidWorks software. These are going to be the elements that you want to pay attention to and get down to ensure you learn modeling well.

COMMANDMANAGER

The CommandManager is like the ribbon of other programs such as AutoCAD® and Microsoft Office® products (Figure 1.3.1).

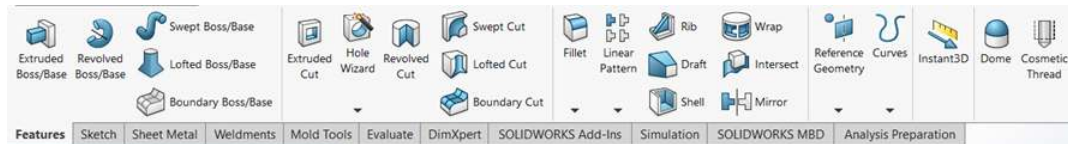


Figure 1.3.1: Copy and Paste Caption here. (Copyright; author via source)

The command manager contains context specific commands based on what type of file you are working in and what you are trying to do. The command manager is composed of tabs that relate to specific functions in SolidWorks. The command manager also looks different based on whether you are creating a new part, assembly, or drawing.

Most of your interaction with SolidWorks will be through the command manager.

The most common CommandManager tabs you will use are the Features and Sketch tabs. To use commands in the CommandManager, you switch to the tab you desire and then left click the command you are looking for. Descriptions will appear next to your cursor when you hover your mouse cursor over a command. When you activate a command by left clicking on it, the property manager will appear with messages and tips on what to do next.

For example, when you left click on the Sketch command in the Sketch tab of the CommandManager, the property manager opens in the manager pane and shows the message, “Select a plane on which to create a sketch for the entity.” The default Front, Right, and Top Planes also highlight and the CommandManager has switched to the Sketch tab (Figure 1.3.2). Interaction with the CommandManager is dynamic in that when you choose a command, things change in the manager pane, CommandManager, and Graphics Area.

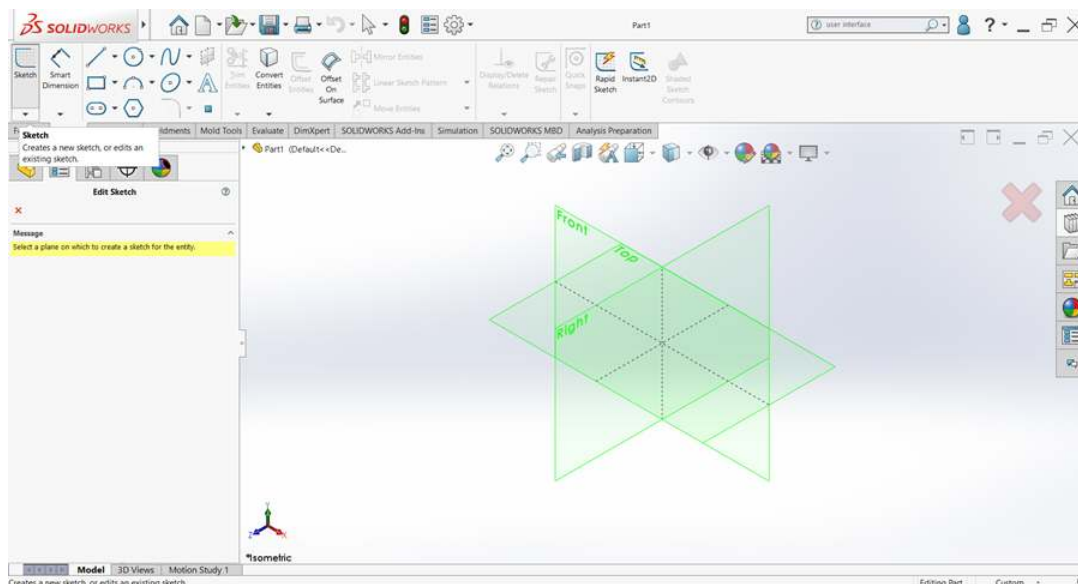


Figure 1.3.2: Example of dynamic nature of the Command manager

GRAPHICS AREA

The [graphics area](#) is where the model is created and where the user interacts with the model. In the graphics area, you can select vertexes (points), edges, faces, planes, axes, features, and bodies associated with a model (Figure 1.3.3).

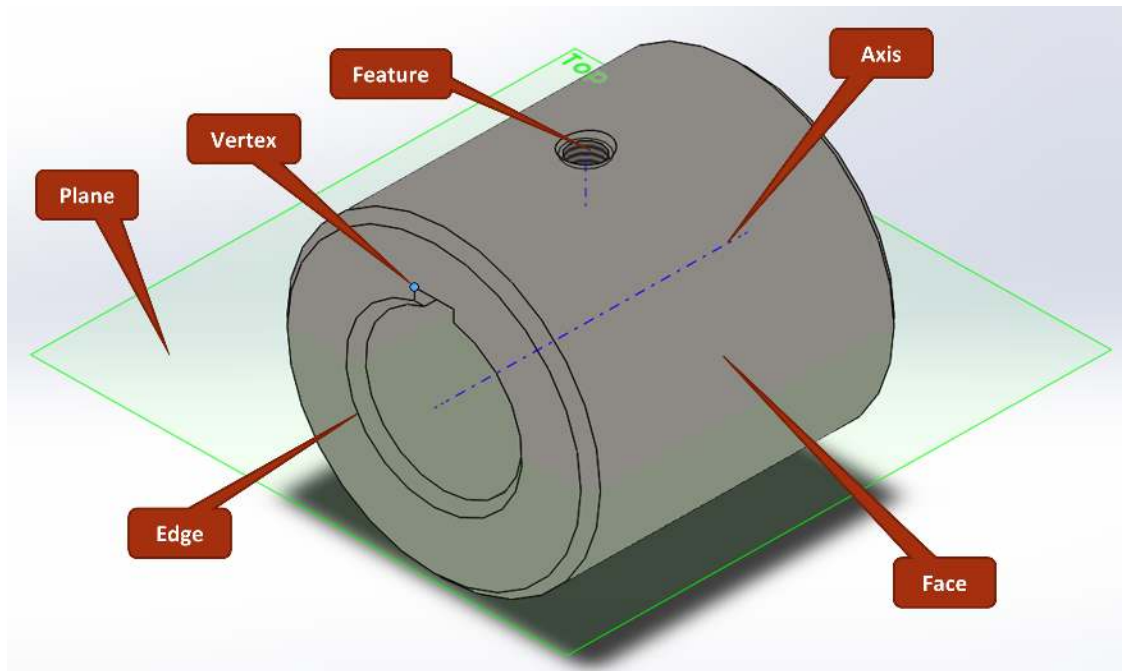


Figure 1.3.3: Selectable Items in the Graphics Area

HEADS-UP VIEW TOOL BAR

The heads-up view toolbar (figure 1.3.4) is right below the CommandManager in the Graphics Area and controls how the model looks in the graphics area. You are able to change the display style, change the view orientation (front, right, top, isometric, etc.), hide/show different items, edit the appearance of items, zoom, and create section views of your model.

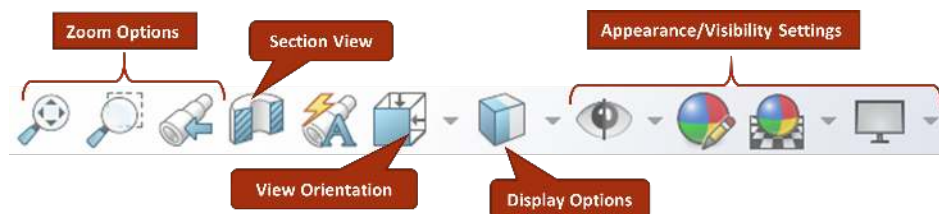


Figure 1.3.4: Heads Up View Toolbar

MANAGER PANE

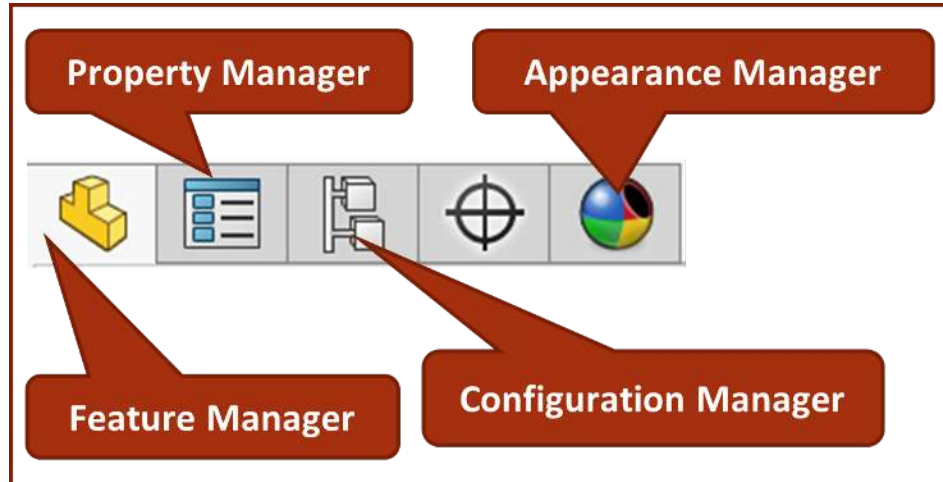


Figure 1.3.5: Manager Pane Tabs

All the tabs can be activated manually. However, the manager pane is dynamic in that it changes to the corresponding tab based on actions you make in SolidWorks®. When a feature or sketch command is activated, the manager pane switches over to the property manager tab; when an appearance command is activated, the manager pane switches to the appearance manager tab. The configuration and dimXpert tabs are only activated when the user manually switches to those tabs by left clicking on them. The feature manager tab is active by default.

FEATURE MANAGER DESIGN TREE

The [feature manager design tree](#) is one of the most important parts of the SolidWorks® user interface. This is the place where features are managed, edited, suppressed, hidden, etc. When you create a new sketch or feature, it appears in the feature manager design tree in the order in which it was created.

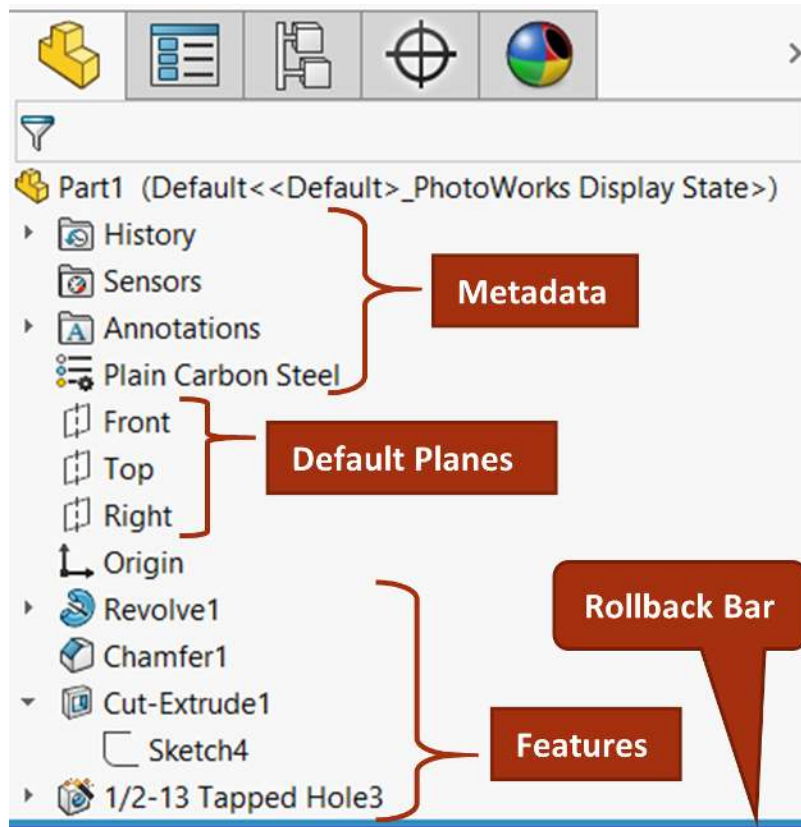


Figure 1.3.6: Feature Manager Design Tree

The feature manager design tree holds the features, sketches, reference geometry, the origin of the part, material designation, annotation folders (relevant only to model based definition), sensors, and history of changes (Figure 1.3.6). The part name always appears at the top of the feature manager design tree along with the active configuration and display state.

The little triangles to the left of the items in the feature manager design tree are called flyouts. You can left click on them to expand or collapse them.

Facts About the Feature Manager Design Tree

- When you left click an item in the feature manager design tree, an in-context toolbar appears next to your cursor with relevant commands.
- When you right click on an item, both the in-context toolbar and a longer list of action options appear.
- Double clicking a folder expands the folder and double clicking a feature shows the feature dimensions.
- Slow double clicking an item allows you to rename the item.
- Doing all these things highlights the respective item in the graphics area (if it exists in the graphics area).

PROPERTY MANAGER

The property manager looks different based off which command is active; even different features have different property managers. The property manager controls the settings of the command that is being executed. For example, if you are extruding a sketch using the Extruded Cut command, you control every aspect of the extrude via the property manager (figure 1.3.7) such as depth (the thickness of your part profile) and draft angle (how much the thickness tapers).

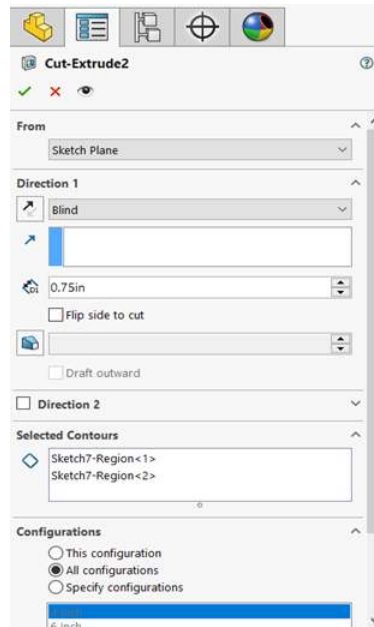
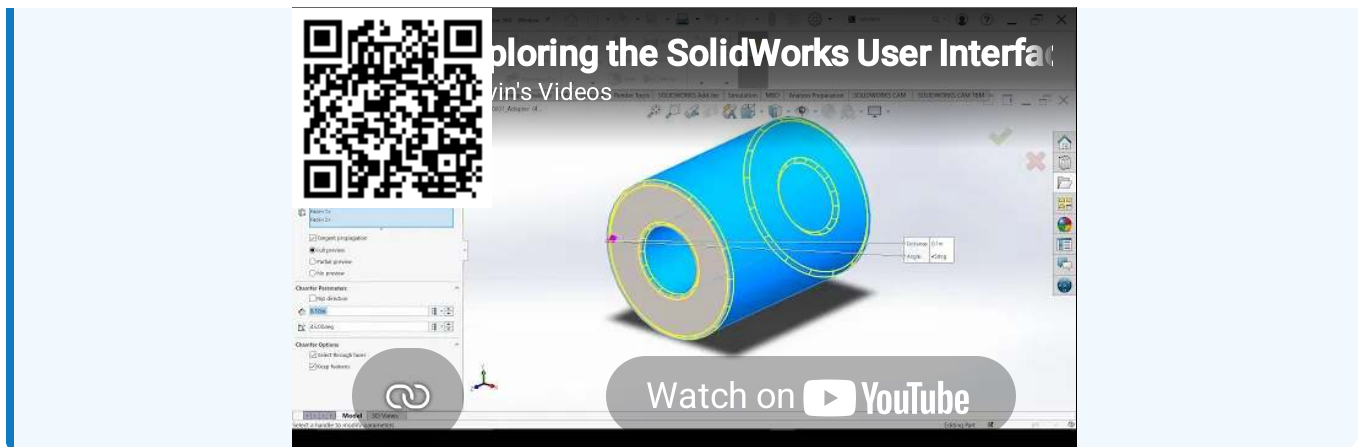


Figure 1.3.7: Property Manager for Cut Extrude

? Exercise 1.3.1: ? Using the Interface to Modify the Model

Within your Coupler part, use the Commandmanager, Manager Pane, and Graphics Area to make the following changes to the Coupler model.

1. Click on the outer face of the Coupler.
2. Change the length to 5 inches and the inner diameter to 1.75 inches by left clicking on the in context dimensions that appear.
3. Click on the Cut-Extrude1 feature in the Feature Manager Design Tree and from the Context toolbar, select the Edit Sketch icon.
4. Sketch a small circle directly below the keyway on the other side of the Coupler inner diameter so that it intersects the Coupler material. Confirm the sketch. There should now be a new small hole in the Coupler part.
5. Click on the Chamfer1 feature in the Feature Manager Design Tree and select the Edit Feature icon from the context toolbar. Change the chamfer size to .1" and confirm the chamfer.
6. From the heads up view toolbar, change the orientation to isometric and create a default section view using the right plane. Then, change the display style to wireframe.
7. Toggle between the different zoom options in the heads up view toolbar.
8. Also from the heads up view toolbar, turn on the temporary axes and primary plane visibility.



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1.4: Other User Interface Components

The remaining SolidWorks User Interface components are non-critical components that communicate the status of the model and allow you to do various other actions with your model such as add supplier components, browse and compare between other SolidWorks files, or navigate to portions of the interface we've already discussed, just in a different way.

STATUS BAR

The status bar at the bottom of the SolidWorks window (figure 1.4.1) shows the status of your model, features, and sketches, coordinates (when in sketches), measurements of selected items, unit settings, and tag and rebuild icon. When sketching, it will tell you the sketch state of the sketch (if the sketch is fully defined, under defined, or over defined). You can change your units and access full unit settings through the status bar.



Figure 1.4.1: Right Portion of the Status Bar

The status bar is useful for gathering quick measurements of edges or coordinate points, adding tags to the model, recognizing whether you are editing a part or assembly, or what may cause an error when you have them, but the most useful feature of the status bar you will find is seeing if a sketch, feature, or assembly is fully defined or not. To capture design intent successfully in a model, all sketches, features, and assemblies need to be fully defined.

All sketches and features should be fully defined!

TASK PANE

The [task pane](#) on the right side of the SolidWorks® window contains resources and add ins for SolidWorks. The task pane doesn't contain any options that impact the model directly; it is more an aid in the modeling process containing several tabs including the [SolidWorks® resources tab](#), [design library tab](#), [file explorer tab](#), [view palette tab](#) (only active for SolidWorks® drawings), [appearances tab](#), [custom property tab](#), and [SolidWorks® forums tab](#). The 3DEXperience Marketplace is also an addition to the task pane since 2020 was released.

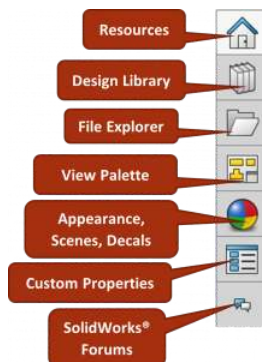


Figure 1.4.2: Task Pane

RESOURCES TAB

The **resources tab** contains resources for modeling, designing, and understanding SolidWorks. It has tutorials, product descriptions of SolidWorks and its add-ins, access to SolidWorks services, as well as other things that are just useful for the modeling process.

DESIGN LIBRARY

The **design library** is exactly as it sounds: a library of design elements that are common in mechanical assemblies such as fasteners, bearings, motors, hubs, etc. The items available in the library are typically common supplier provided parts for

manufacturing companies. You can add things to the library as well such as commonly used weldments, assemblies, custom fasteners, structural steel cross sections, etc.

FILE EXPLORER

The **file explorer** tab allows you to access files straight from SolidWorks without needing to open the Windows® file explorer. An advantage to this is being able to preview all SolidWorks documents (say you forget what you named something) and rename/move SolidWorks documents without messing up the file links (such as links between parts and assemblies/drawings; see page “[File Management with SolidWorks Files](#)” from earlier in the chapter).

VIEW PALETTE

The **view palette** is a tab only available when in a SolidWorks® drawing and is used to place 2D drawing views in a drawing. When you start a drawing from a SolidWorks part, the View Palette opens by default for you to left click drag and drop orthographic drawing views onto a drawing sheet.

APPEARANCES TAB

The **appearance, scenes, and decals tab** allow you to change the appearance, scene (background), and add decals to a part/assembly without changing your view. There are many other ways to change the appearance, scene, and add decals (through the Appearance tab in the manager pane, the Heads-Up View Toolbar, and through context toolbars).

CUSTOM PROPERTY TAB

The **custom property tab** allows you to quickly specify custom properties for the model. Custom properties are often the metadata of the SolidWorks® file (see “[File Commands Bar](#)” earlier in the chapter). Assigning metadata to design files is important for use downstream the **product life cycle**. The custom property tab shows options based on a custom property tab that was built in the [custom property tab builder](#). Throughout this book, you will use a custom property tab that is provided.

SOLIDWORKS FORUMS TAB

The [SolidWorks® Forums tab](#) allows you to access the online forums straight from SolidWorks®. Forums are where users of the software post questions and answers related to issues, they find using the software. The SolidWorks® forums are filled with tips and tricks around modeling and assembly problems. When you run into an issue in the software, it is likely you will find yourself searching the web from forum to forum. Now knowing about the task pane tab, don’t waste your time switching between SolidWorks and web browser; just search within SolidWorks!

3DEXPERIENCE MARKETPLACE

3DEXperience marketplace was added in SolidWorks 2021 with the role out of 3DEXperience as an integrated product lifecycle ecosystem. SolidWorks can now integrate directly into the platform. 3DEXperience marketplace allows you to browse and collaborate suppliers across the globe for standard component parts.

CONTEXT TOOLBARS

Context toolbars appear when you left or right click on a model item in the graphics area or on an item in the feature manager design tree. These toolbars contain common and helpful commands related to whatever you selected.

For example, if you click on a sketch in the feature manager design tree, the context toolbar will contain many common commands for sketches. You will use these mostly for editing sketches and features.

Context Toolbar Tip

The context toolbars only appear when you need them, so as soon as you click on an object they appear; if you move your mouse cursor away from the toolbar, it disappears, but if you move your mouse cursor towards the toolbar, it stays visible for you to interact with the commands in the toolbar.

? Chapter Review and Reflect

In this chapter, we were introduced to what SolidWorks is and how users interact with the SolidWorks software through the graphical user interface. We explored some general elements that are found in the top portion of the window, the major components such as the Commandmanager, the Graphics area, and the Manager Pane, and other components that help users be more efficient in their mechanical design practice. To solidify your learning of the material presented here, reflect through the following review questions.

- What type of CAD software is SolidWorks?
- What types of files are used with SolidWorks?
- How alike is SolidWorks to other programs?
- How are models modified using the User Interface?
- What new glossary terms was I exposed to in this chapter?
- How will what I've learned here help me achieve my career goals?

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CHAPTER OVERVIEW

2: Sketching and Relations

Learning Objectives

To create simple parts, we need to first learn how to do the following:

- Identify sketch planes and sketch faces.
- Create new sketches using Smart Dimension, geometric relations, and other sketch tools.
- Create new Extruded features from sketches.
- Differentiate between sketch-driven features and feature-driven features.

Now that we have great exposure to the SolidWorks User Interface, we can begin learning how to create simple parts. For the purposes of this book, we will define simple parts as those only having one or two features in the FeatureManager Design Tree.

[2.1: Starting a Sketch](#)

[2.2: Sketch Creation Tools](#)

[2.3: Smart Dimension](#)

[2.4: Modification Sketch Tools](#)

[2.5: Geometric \(Sketch\) Relations](#)

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2.1: Starting a Sketch

When you open SolidWorks and start a new part, you will notice in the FeatureManager Design Tree a Front, Top, and Right default Plane (Figure 2.1.1).

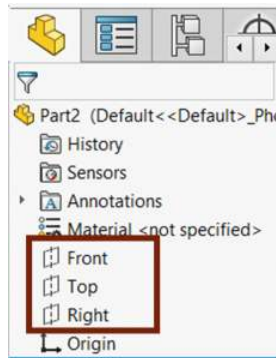


Figure 2.1.1: Front, Top, and Right Default Planes

These are hidden by default, but in figure 2.1.2, they are shown to illustrate the orientation of the default planes. The default planes all intersect to create an origin. One of these planes is usually the starting point for any part model. You start sketching on the plane that is best oriented to the orientation you wish to start creating your part.

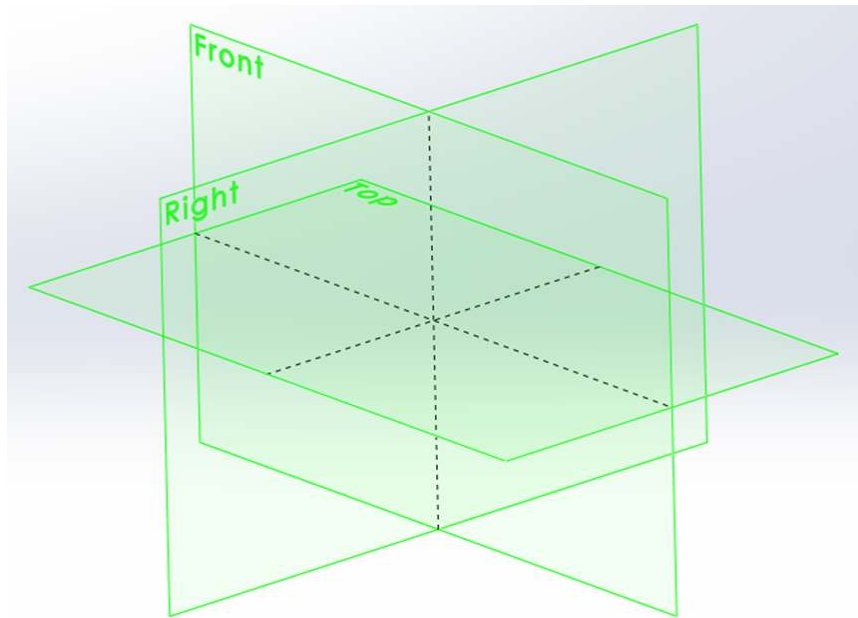


Figure 2.1.2 Front, Top, and Right Default Planes Shown in Graphics Area

SKETCHING ON PLANES

When starting a new part, you want to begin your first sketch on the plane that will show the most profile and detail of your part. If the most profile and detail exist in the front or back view for your part (such as for a pipe), then you should consider beginning on the front plane (Figure 2.1.3). If the most profile and detail exist in the top or bottom view of your part (such as for a base plate), then you should consider beginning from the top plane (Figure 2.1.4). If the most profile and detail exist in the right or left view of your part (such as for a wheel hub), then you should consider beginning from the right plane **Figure(2.1.5)**.

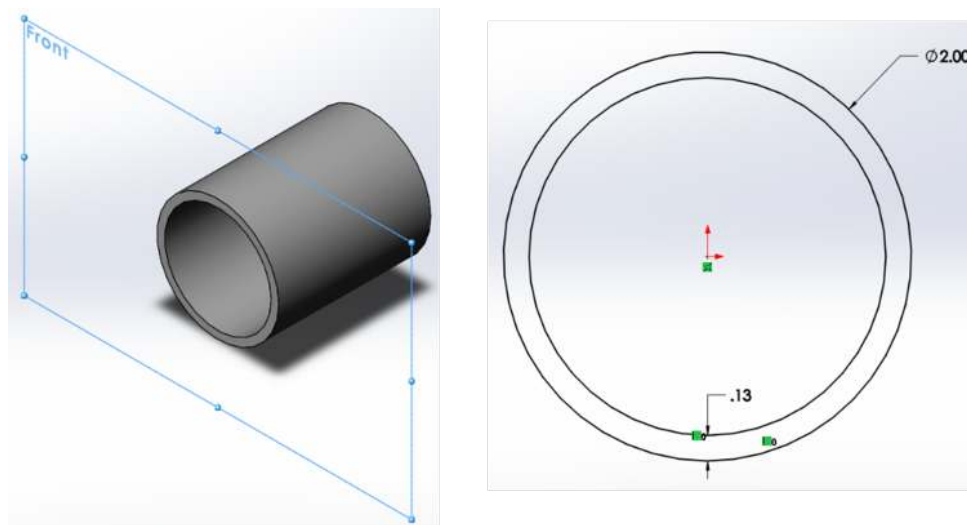


Figure 2.1.3: Sketching from Front Plane

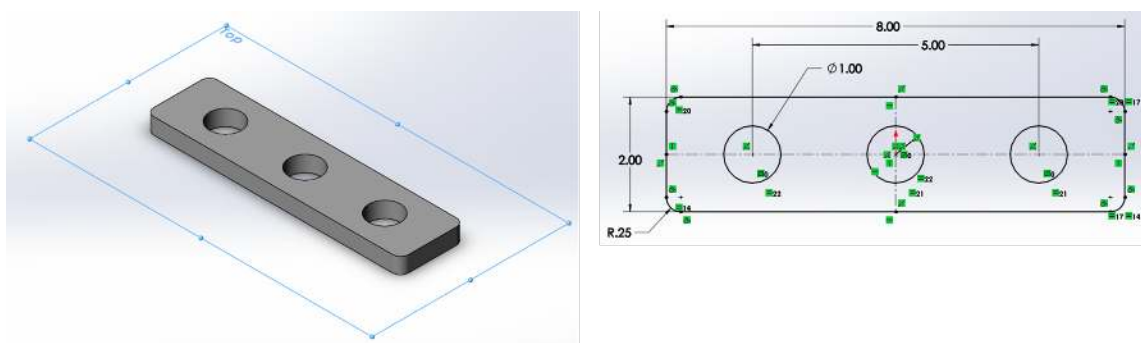


Figure 2.1.4: Sketching from Top Plane

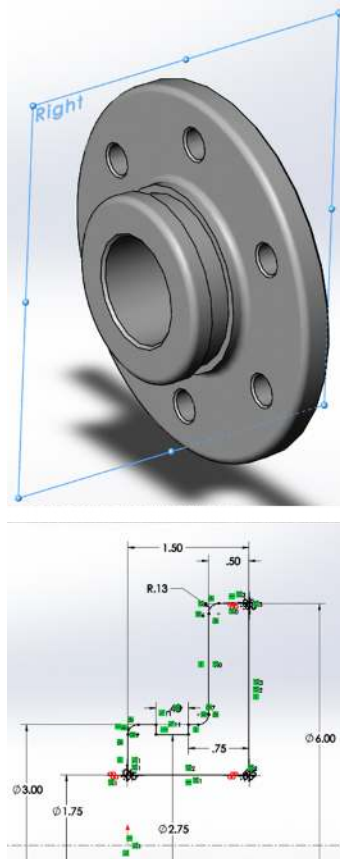


Figure 2.1.5: Sketching from Right Plane

STARTING A SKETCH FROM A DEFAULT PLANE

A sketch can be started from a default plane in a couple of ways such as from the menu bar and clicking on the “Sketch” button from the sketch tab and following the property manager directions. However, the quickest way to start a new sketch from a default plane, and really any plane or face, is to left or right click on the desired plane (in the FeatureManager Design Tree or Graphics Area) and select the “Sketch” Icon (Figure 2.1.6). This puts you right into the sketch environment.



Figure 2.1.6: Sketch Icon

The sketch environment is to the SolidWorks User Interface like the inside of a house is to the rest of the outside world. Going into the Sketch Environment makes many features off limits; however, you have access to so many other new features in the sketch environment related to creating sketches. Just like the things you do inside a house are completely different from the things you do outside, the things you do inside the sketch environment are completely different from the things you would do outside the sketch environment.

Note: Many new users to SolidWorks frequently go into the sketch environment by accident without knowing it, and then are confused when they find out that they are no longer able to complete the commands they are usually able to complete. If you find yourself unable to do things in SolidWorks you should be able to do, check to see if you are in the Sketch Environment.

SIGNS THAT YOU ARE IN THE SKETCH ENVIRONMENT

The **first sign** to know if you are in a sketch is to check the status bar. The status bar in the right-side portion should say “Editing Sketch...” for whatever sketch you are editing (Figure 2.1.7).



Figure 2.1.7: Sketch Editing Status from Status Bar

The **second sign** is the in-context confirm and deny buttons in the upper right-hand corner of the Graphics Area (Figure 2.1.8). The blue arrow button is to confirm changes made to a sketch; the red “X” is to deny, and discard changes made to a sketch.



Figure 2.1.8: Sketch Confirmation/Discard Buttons

The **third sign** is your CommandManager changing to the Sketch tab and the Sketch button in the CommandManager being indented a dark grey shade (Figure 2.1.9). Another, less noticeable sign, is your FeatureManager Design Tree features becoming greyed out. Under any other circumstances, you are **NOT** in the sketch environment and **CANNOT** do sketch related commands.

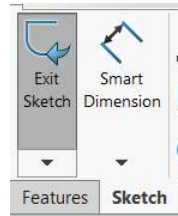


Figure 2.1.9: Indented Sketch Button when Inside a Sketch

? Exercise 2.1.1? : Going In and Out of Sketches

Let’s take some time to practice going in and out of the sketch environment and recognizing the various signs that we are in or not in the sketch environment.

- Open a SolidWorks new part.
- Enter and exit the sketch environment in the ways described above.
- Each time you are in the sketch environment, look for the three signs that indicate you are in the sketch environment.



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2.2: Sketch Creation Tools

The sketch tools within SolidWorks® are very similar to those in other **CAD** and Parametric programs: line, circle, rectangle, polygon, slot, arc, etc. (Figure 2.10). Each command also has drop down options to allow you to have more control over how you specify a geometric element in a sketch. There are yet even more customizing options for each sketch tool in the property manager for that sketch tool.

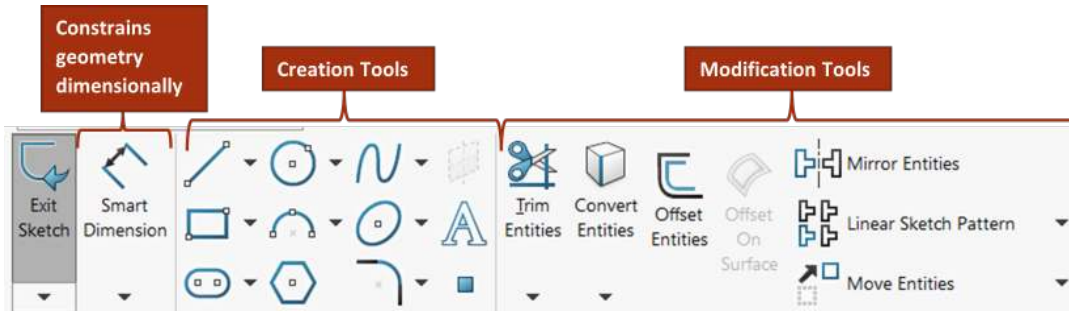


Figure 2.10 Primary Sketch Tools

As soon as you activate a sketch tool by left clicking on it from the **CommandManager**, the property manager for that sketch tool opens, usually with a set of illustrative directions for how to use the sketch tool.

These tools are going to be the most straightforward tools because they work the same across all platforms (for the most part). However, as you add sketch geometry to a sketch, often times **sketch relations** will be automatically added to the origin and the entities you are adding. We will discuss these more later in this chapter and how to use them but for now just be aware that you will see them appear on your sketch as green tiny boxes.

DESIGN INTENT IN SKETCHING

SolidWorks called the Sketch tools “Sketch” for a reason: the intention behind parametric design is to do a rough sketch of what you want first, and then constrain it using dimensional (Smart Dimension) and geometric (Relations) constraints. This will allow you to model way faster than if you tried to get all the details right as you sketched. Sketching this way also ensures that you are modeling based on **design intent**, not putting any unnecessary constraints on the geometry that are not needed for the design to function. An example is shown below in figures 2.11, 2.12, and 2.13.

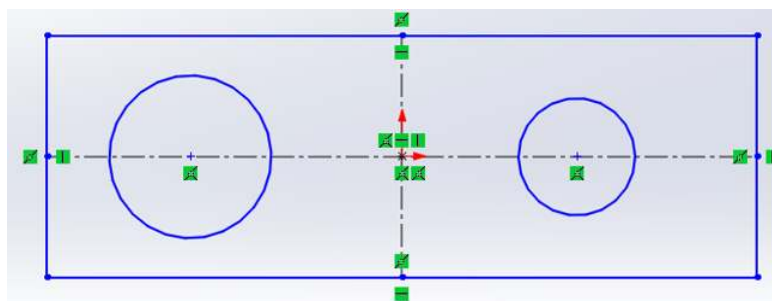


Figure 2.11 Create rough sketch without adding constraints

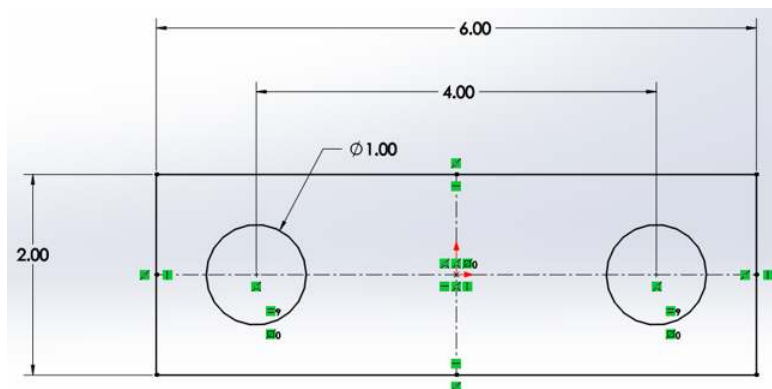


Figure 2.12 Add dimensional and geometric constraints using smart dimension and relations

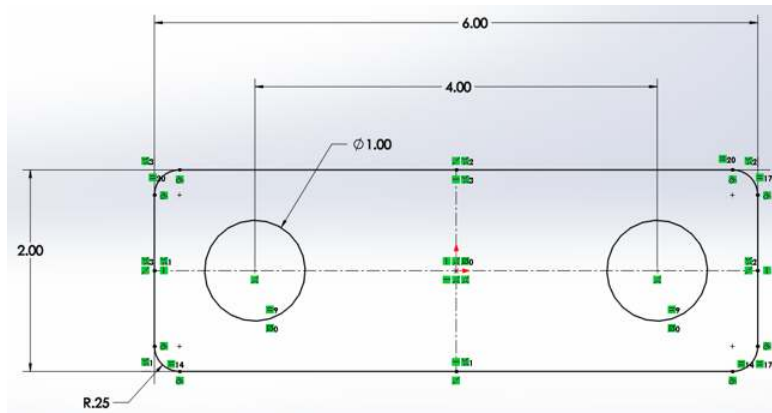


Figure 2.13 Add entity-based features such as fillets and *chamferse*)

You will notice that the sketch in figure 2.11, 12, and 13 started out as blue and then, constraints were added, it turned black. When features/entities are blue in the sketch environment, they are **under-defined**. When features/entities are black in the sketch environment, they are **fully defined**. These states are known as **sketch geometry status**.

When a sketch is fully defined, all the appropriate design criteria needed to manufacture the feature(s) has been specified by the designer. When a sketch is under defined, this means that there is not enough information, not enough design criteria specified in the sketch environment to manufacture the features drawn.

Look at Figure 2.11 again: without the dimensions on the geometry to specify the intended size and location for the drawing, how would manufacturing know what the design intent was for the part? Without the symmetry constraint on the two circles between the vertical center line, how would manufacturing know where one hole is supposed to be with respect to the rest of the part? Yes, the information is in the model, but there is variation in manufacturing processes no matter what process it is, so what do we compare the manufactured parts to if there is no design information in the model? Therefore, it is critical to make sketches fully defined before sending them downstream in the product design process.

Fully define all your sketches!

LINE

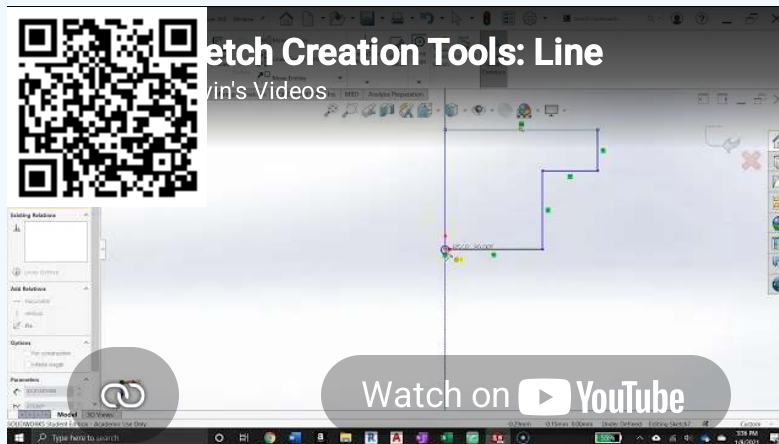
When you activate a tool, such as the line tool, you specify the location of the base point with a left click and then the size and orientation with the second left click; to exit the line command, simply right click and select "Select" or click the green check mark (Figure 2.14). Many sketch creation tools have expanded options. The line command has a midpoint option where you can specify the midpoint of the line on your first mouse click as opposed to the first endpoint.



Figure 2.14 Creation of a Line

? Exercise 2.2: Using Line

In this exercise, let's take some time to practice the line sketch creation tool, study its property manager, and experiment with the different options.



RECTANGLE AND CIRCLE

The rectangle tool also allows you to either specify rectangles using two corners, the center of the rectangle, or 3 corner points. Below is an example of using the center point of the rectangle as the base point:

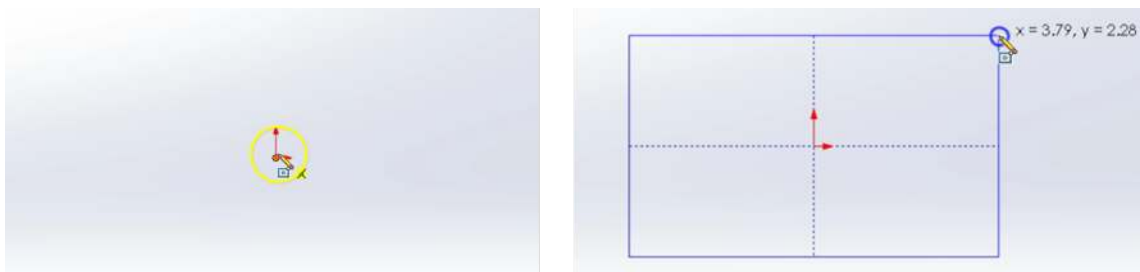


Figure 2.15 Creation of a center-rectangle with midpoint lines

The property managers of all the sketch creation tools have many options and instructions on how to use each option. All sketch tools allow you to convert the sketch entities into **construction geometry** (also known as reference geometry). The rectangle tool specifically has different options for how the construction lines that help constrain the rectangle are placed: at the corners or at the midpoints (Figure 2.16).

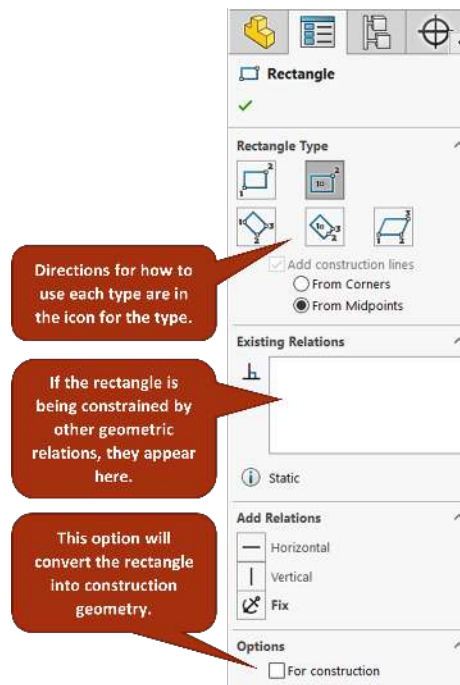


Figure 2.16 Rectangle Sketch Tool Property Manager

In Figure 2.16, you can see that in the rectangle type icons, there are number 1, 2, 3. These indicate the order in which you left click to specify those points indicated in the icons.

? Exercise 2.3: Using Rectangle and Circle Sketch Tools

In this exercise we will practice using the rectangle and circle sketch tools, exploring their property managers, and experimenting with the different options.

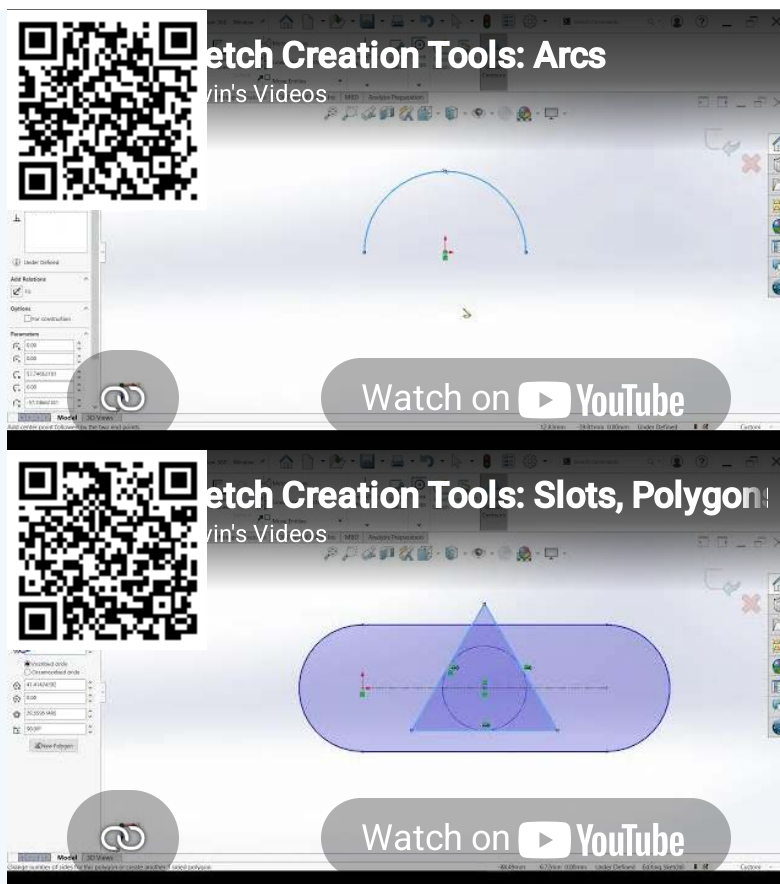


ARCS, SLOTS, POLYGONS AND ELLIPSES

The other sketch commands work in a very similar fashion as the line and rectangle commands. It is important to practice each sketch command, study each property manager for each sketch command, and try different variations of using the sketch commands to build proficiency in them.

? Exercise 2.4: Arcs, Slots, Polygons and Ellipses

Slots, polygons, and ellipses involve a bit more action with their commands to get them into your sketch. They still have similar behavior to the sketch creation tools we have just explored though. Spend some time practicing these sketch creation tools.



FILLET AND CHAMFER

Entity based sketch tools such as fillet and chamfer allow you to quickly add complex but common sketch entities by modifying sketch entities you have already created. For example, if you've already drawn a rectangle using the rectangle tool, you can activate the sketch fillet tool, specify a size in the property manager, then click the corners you want to add fillets to (Figure 2.19).

It is best to use these tools after the sketch is already fully defined.

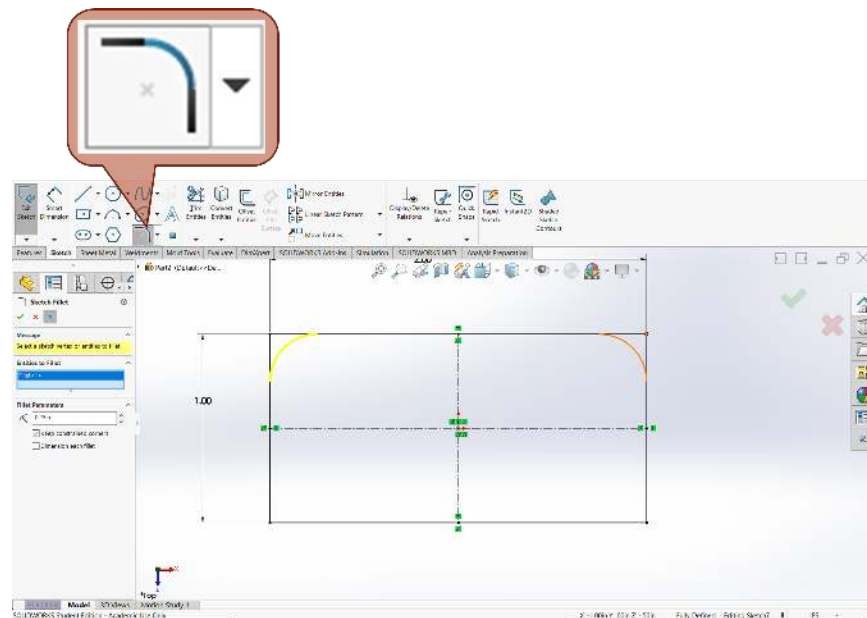


Figure 2.17 Fillet Sketch Tool

? Exercise 2.5 Fillets and Chamfers

Fillets and chamfers are a sometimes useful entity-based sketch creation tool especially when there are other features that need to be added that are related to the fillet or chamfer. Let's practice using the fillet and chamfer tool to become familiar with how it works.



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2.3: Smart Dimension

The smart dimension tool (Figure 2.3.1) has many options in its drop down; however, the intuitive nature of the general smart dimension tool usually suffices for all sketch dimensioning scenarios. You activate the Smart Dimension tool by clicking on it in the CommandManager (Sketch Tab), then you click what you want to dimension.



Figure 2.3.1: Smart Dimension Tool

? Exercise 2.3.1 Exploring Smart Dimension

Instead of describing every way to use the smart dimension tool, let's test out and experiment with the ways we can use the smart dimension tool to discover its intuitive nature.

- Clicking one sketch entity will dimension a length of a line, diameter of a circle, or radius of an arc.
- Clicking on two sketch entities will dimension the distance between two lines, two circles, two arcs, two points, or a combination of these.
- Clicking a center line and a line parallel to it, or a point, will dimension twice the distance between the entity and the centerline to simulate a diameter or symmetry that isn't yet created.
- Clicking on two nonparallel lines will dimension an angle.
- Clicking on the end points of an arc and the curve of the arc will dimension arc length.

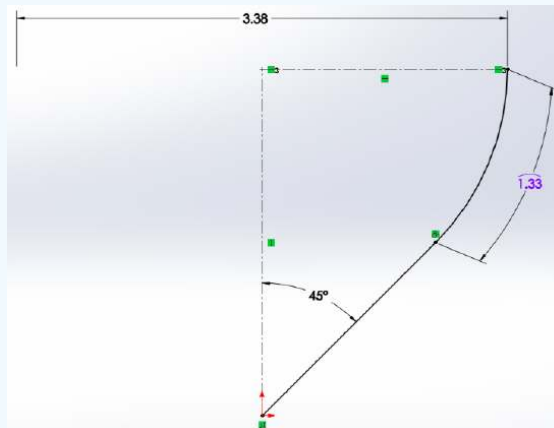
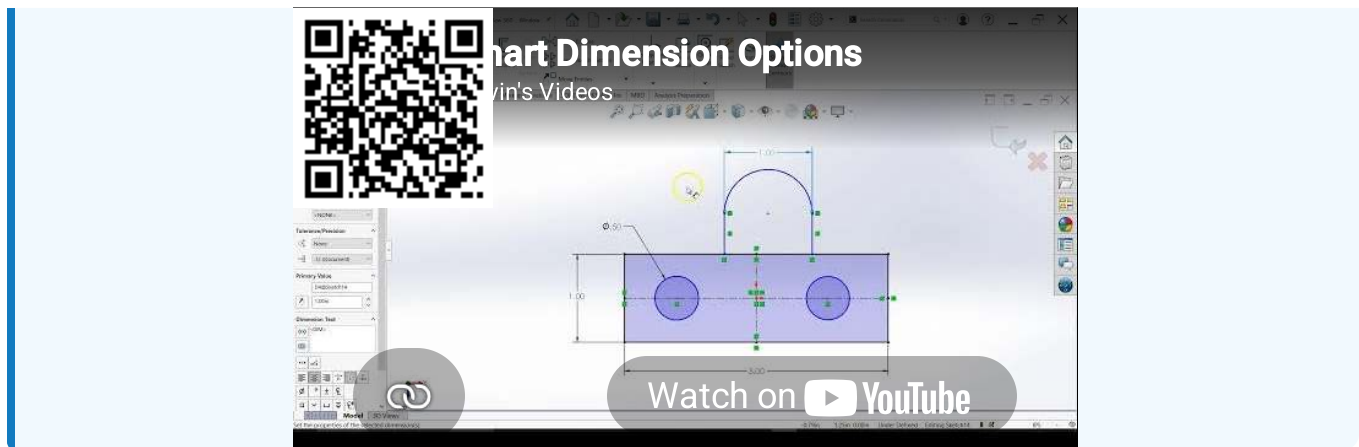


Figure 2.3.2: Dimensioning Angles, Arc Lengths, and Diameters/Symmetry



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2.4: Modification Sketch Tools

The modification sketch tools are great tools for making sketching more streamlined and efficient. Everything that these tools do can be done in other ways but using these tools can be so much faster. Most these tools look familiar if you have ever been in a CAD software before such as AutoCAD. You won't be surprised to find that the function of these tools in SolidWorks is very similar to that in other programs. One tool that probably doesn't look familiar is Convert Entities. We will cover this tool in the simple extrudes part.



Figure 2.20 Modification Sketch Tools

TRIM ENTITIES

Trim entities is used to erase sketch elements that are between another element. A great example is the sketch shown below. You can use a rectangle and a line to create a T-shape such as is shown, but what you don't usually want is the line that separates the rectangle from the rest of the sketch. **Power Trim**, the most common use of Trim Entities, can be used by left click dragging your mouse cursor through the lines you wish to remove (Figure 2.21). In using power trim, you should make sure not to drag your mouse cursor through entities you DON'T want to remove. Trim Entities also has an **Extend Entities** option in its drop down menu.

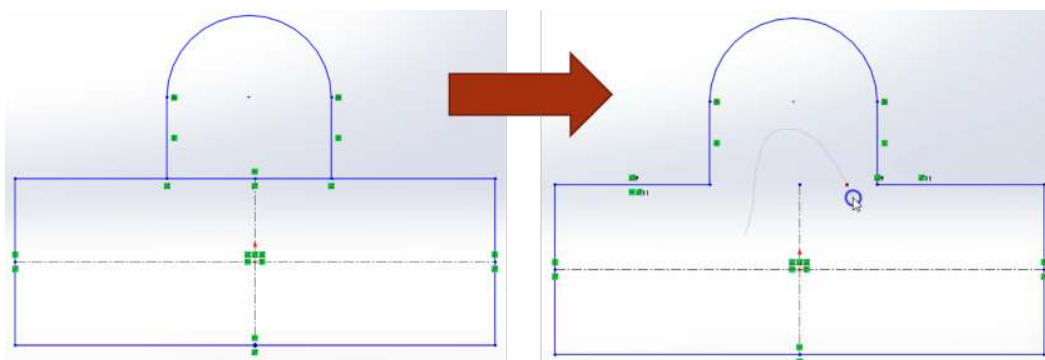
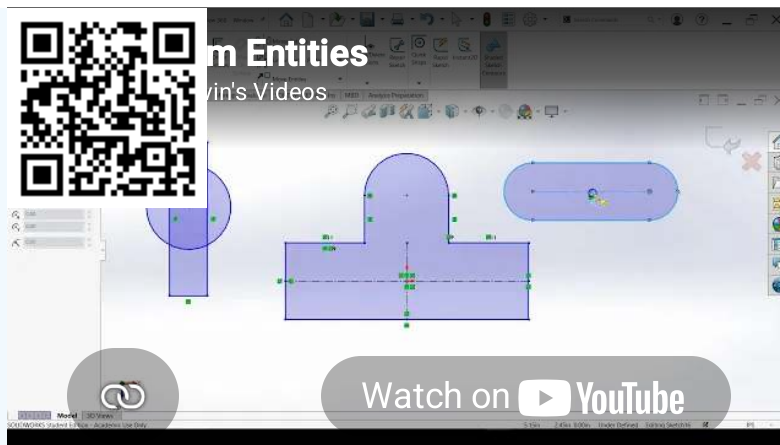


Figure 2.21 Using Power Trim

? Exercise 2.7: Trim Entities Practice

Practice using the trim command in various geometric circumstances. There is a single tutorial below but feel encouraged to sketch more scenarios in which you can practice trim. Don't worry about breaking SolidWorks; you can always discard the sketch and ask your instructor about it later.

- Sketch the geometry above on a default plane.
- Activate the trim tool and power trim through the shown line segments.



OFFSET ENTITIES

Offset entities work exactly as it sounds: you select sketch entities to copy in a chosen direction by a certain distance. This tool works similar to other CAD programs but has some enhanced capabilities such as the ability to offset in two directions at once, automatically trim connected entities based on design intent, add caps on the ends of offset entities to close them, and convert base or offset geometry into **construction geometry** (Figure 2.22).

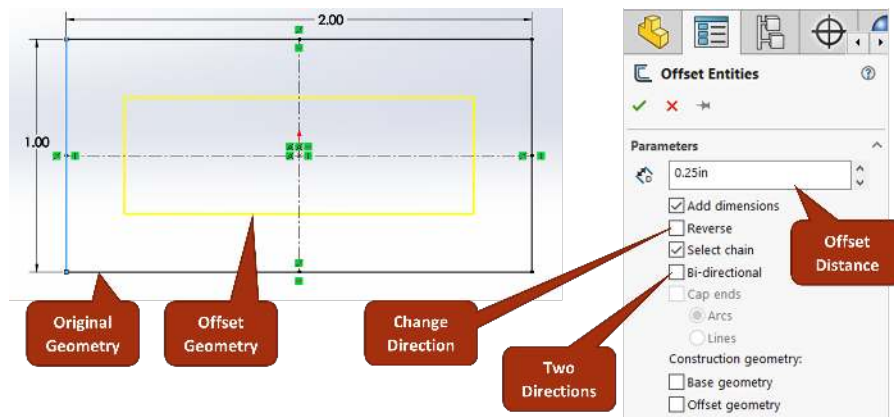
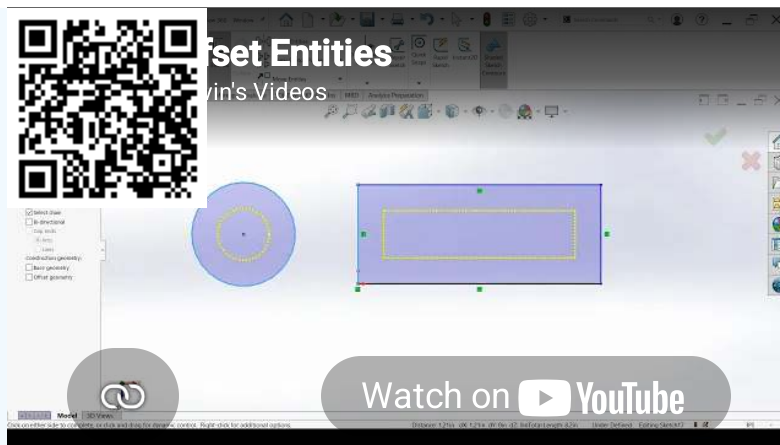


Figure 2.22 Offset Entities Example

? Exercise 2.8: Offset Entities Practice

Now let's practice using the offset entities command. Just as we did with trim entities, make lots of different types of geometry and practicing offsetting and using the different property manager options with the offset.

- Sketch out some various sketch geometries in SolidWorks.
- Use the Offset Entities command to test out different ways to offset geometry.



MIRROR ENTITIES

Mirror entities creates a copy of something as well as flips the orientation based on a mirror line; it's as if those entities were looking in a mirror. Using mirror entities first requires that you draw the entities you want to mirror as well as having a line to represent your mirror line. The mirror line can be either a solid or construction line, but it is usually a construction line.

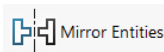


Figure 2.23 Mirror Entities

Creating a Construction Line

- Select “center line” from the line drop down when in a sketch (Figure 2.24).

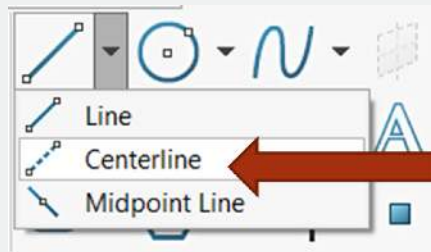


Figure 2.24 Construction Line from Line Drop Down

- Click on a solid line and select the “Construction Geometry” Icon from the context toolbar (Figure 2.25).

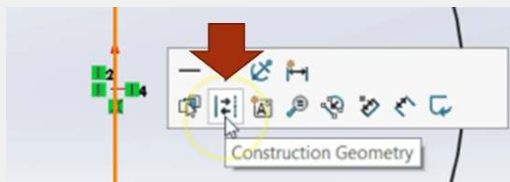


Figure 2.25 Construction Line from In Context Toolbar

- Select a line and then check the box in the property manager that says, “For Construction” (Figure 2.26).



Figure 2.26 Construction Line from Property Manager

To use the mirror tool, select it from the CommandManager, select the entities you wish to mirror; then change to the mirror line selection box by clicking inside it from the property manager, then select your mirror line. Confirm the command in the property manager to confirm the mirror operation.

? Exercise 2.9: Mirror Entities Practice

Practice drawing construction lines and using the mirror entities tool. The mirror entities tool doesn't have many options because it is really just a shortcut for using symmetry relations to copy and constrain your geometry.

- First create a new sketch with some geometry to be mirrored.
- Create a construction line to represent the mirror line.
- Use the mirror entities tool to mirror your geometry.



LINEAR AND CIRCULAR SKETCH PATTERN

Linear sketch pattern is used less often due to the common preference amongst designers to use the linear/circular pattern feature which is outside the sketch environment (more on this in chapter 3). However, the sketch pattern comes in handy for special situations. To use the sketch pattern feature, you must create the geometry you wish to pattern first before activating the sketch pattern tool. Once you have the geometry, for instance a circle, you can use the sketch pattern property manager to specify the number of instances you'd like, the direction (horizontal/vertical direction), and the spacing (Figure 2.27). **Circular sketch pattern** is in the drop down for linear sketch pattern where you select a point as a center for the pattern and pattern radially.

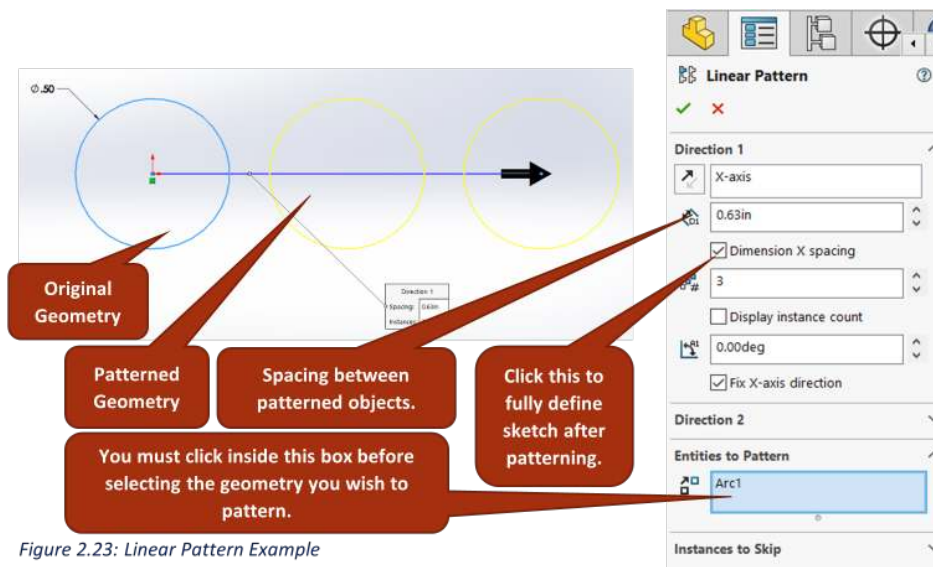


Figure 2.23: Linear Pattern Example

Figure 2.27 Linear Sketch Pattern

? Exercise 2.10: Linear and Circular Sketch Pattern

Linear and circular sketch pattern are a bit trickier to use so close attention should be paid here when practicing its use. Watch the video tutorial closely below and then experiment on your own with using the tool with all its property manager options.



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2.5: Geometric (Sketch) Relations

Geometric relations (also known as sketch relations) are constraints that you place on a sketch to specify design criteria that isn't dimensional. These constraints are things like being on center, square, symmetric, in line, equal, etc. You will recognize geometric relations as the little green boxes you see in a sketch. Each geometric relation has its own icon indicating what it is.

Geometric relations are added by activating the “**Add Relation**” command in the Sketch tab of the CommandManager (Figure 2.28).

You can also hold CTRL on your keyboard and select entities and relate them using the in context toolbar.

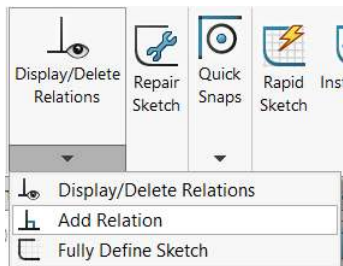


Figure 2.28 Add Relations Command from CommandManager

As you select entities in the Add Relation command, different relations automatically become available for selection in the property manager for the command based on what you are selecting (Figure 2.29). You can also see that SolidWorks will suggest a relation for you to use (in bold font) based on your current design and what you have selected.

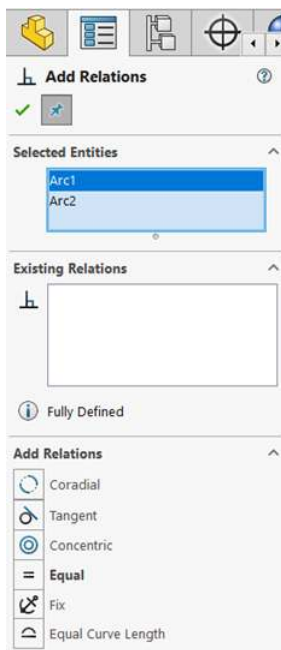


Figure 2.29 Add Relation Property Manager

You can also delete relations from sketch entities' property manager (every entity has a relations section of their property manager) or by simply left clicking the green relation box, right clicking and selecting “Delete” (the delete key on the keyboard also works).

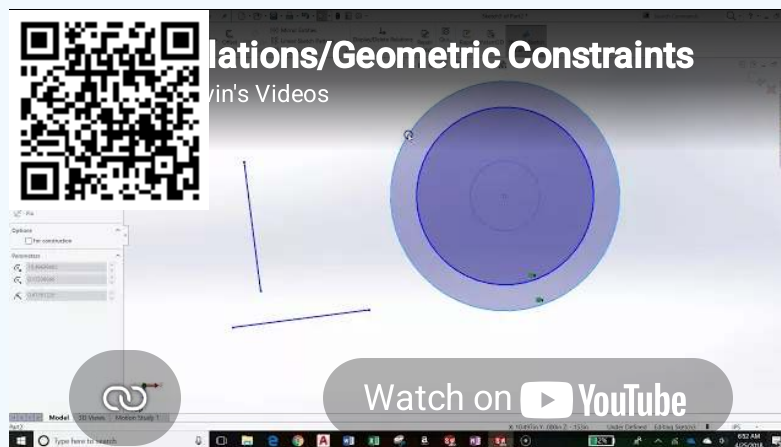
There are so many geometric relations as can be seen from following the link attached to “Geometric Relations” at the beginning of this paragraph. You can see what the icons look like and what it looks like when the geometric relations are applied through

[SolidWorks Help: Geometric Relation Icons](#). Many of them are inferred through other commands and many of them are only used for special cases; we will observe those most used:

- | | |
|---|---|
| <ul style="list-style-type: none">• Horizontal• Colinear• Coradial• Perpendicular• Parallel• Tangent | <ul style="list-style-type: none">• Vertical• Concentric• Midpoint• Coincident• Equal• Symmetric |
|---|---|

? Exercise 2.11: Exploring Geometric Relations

Spend some time experimenting on your own with the different relations you can add to entities. Use the video below as a reference. Note you can quickly add relations by holding CTRL on your keyboard and selecting entities, choosing relations from the in-context toolbar.



Note: Because many relations are inferred, new users of SolidWorks often find themselves sketching and getting stuck on a sketch due to accidentally adding an inferred relation that isn't desired. If you ever find yourself trying to add a dimension or relation to a sketch but keep getting over-defined warnings, check the sketch for any unwanted relations.

CLEVIS PLATE EXAMPLE

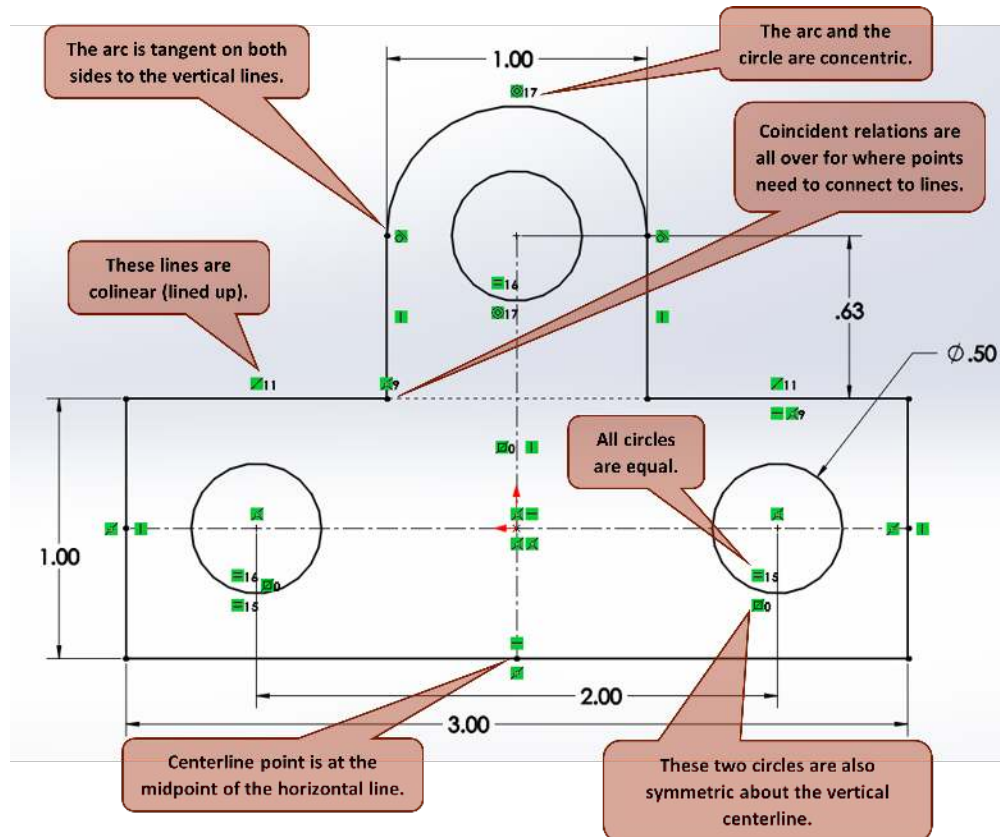


Figure 2.30 Sketch Relations Example Clevis Plate

“Merge points” is a common option in the “Add Relation” property manager but it doesn’t have a little green box to go with it. The upper point of the vertical center line was merged with the center point of the upper circle and arc. Therefore, there aren’t any relations near that point.

Something else you might notice is the greyish dashed line connecting the upper left line to the upper right line. This dashed line is showing that the two solid lines are colinear. This can be helpful in more complicated sketches.

To create the symmetric relation, both circles and the vertical centerline had to be selected. When making the tangent relations, you want to make sure that when you select the tangent arc and line, you are clicking near the desired point of tangency. Not doing this can lead to an unexpected tangent result.

Some things you will notice about the image above is that there are a lot of coincident relations. Coincident relations are usually added by SolidWorks automatically when you begin a base point on an entity that is already in the sketch. You should also observe that coincident relations look a lot like midpoint relations, so to be sure of which is which, you can hover over the relation to get the pop-up identifier.

CYLINDER GASKET EXAMPLE

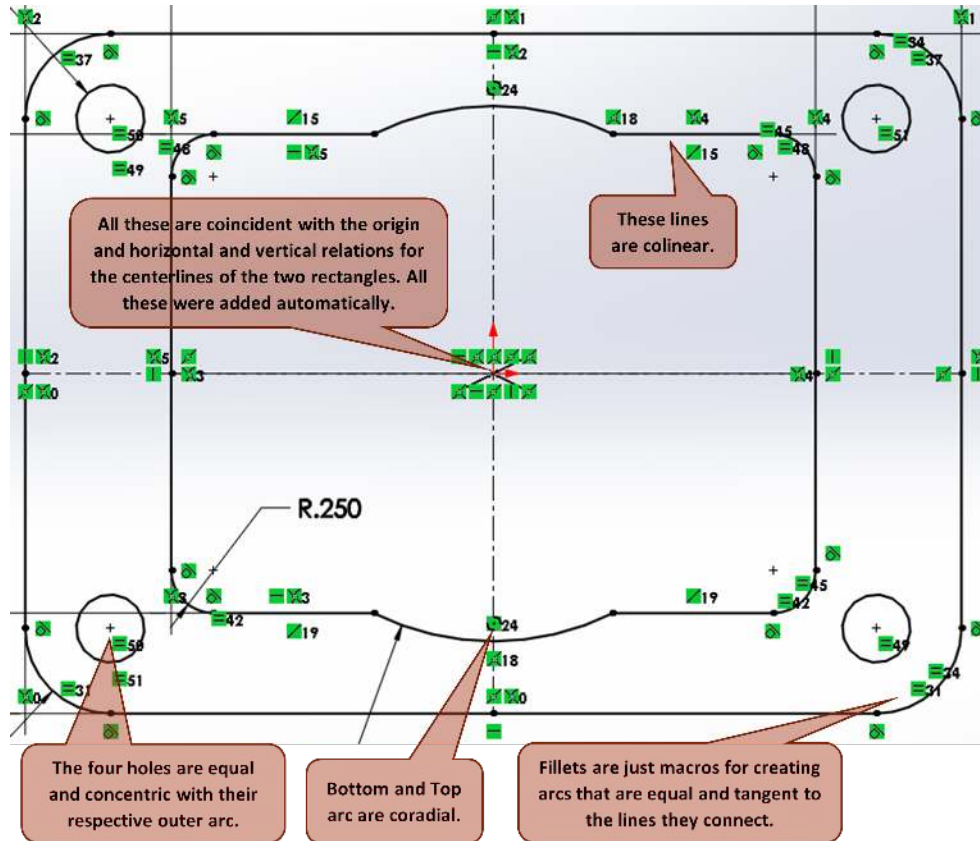


Figure 2.31 Sketch Relations Example Cylinder Gasket

To create this sketch, the two center rectangles and a circle placed at the origin to the diameter of the middle arcs shown were drawn. Power trim was used to leave just the upper and lower center arcs. A quick way to fully define the four holes at the corners were to create the fillets before the holes so that the center of the holes could be placed on the center point of the fillets, resulting in automatic merging of the points.

You will notice that when the fillet tool is used, it just makes arcs and adds a lot of relations. For this reason, heavy use of the sketch fillet or chamfer tool can lead to slower running of the software. This is primary argument for why many designers choose to stay away from the sketch fillet/chamfer tool altogether and use the feature fillet/chamfer tool instead. The same also goes for linear and circular sketch patterns. The feature versions of these tools are much less taxing on the software. These are tools we will learn in future chapters.

Make sure to spend LOTS of time practicing with sketch relations. They are one of the most important aspects of SolidWorks!

SIMPLE EXTRUDES

When we create a sketch, the purpose is to then apply a feature to it to make the 2D sketch 3D. The most common feature used to make a sketch into a 3D feature is the **Extruded Boss/Base** command on the CommandManager.

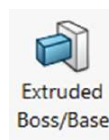


Figure 2.32 Extruded Boss/Base Icon

You can apply a feature to a sketch inside the sketch environment or outside the sketch environment. When inside the sketch environment, when you have finished the sketch, switch to the **features** tab of the CommandManager and click on Extruded Boss/Base. When outside of the sketch environment, select the sketch you wish to extrude from the FeatureManager Design Tree (it will become highlighted) and then click on the Extruded Boss/Base tool on the CommandManager. These techniques apply to all single sketch features.

Once the Extrude tool has been activated, the property manager will open, and a yellow preview of the feature will appear (Figure 2.33). The property manager is where you will specify the depth of the extruded feature (thickness you will apply to your zero-thickness sketch). There are other options as well to help with design intent and to give you more control over your feature. We will explore these more in the next chapter.

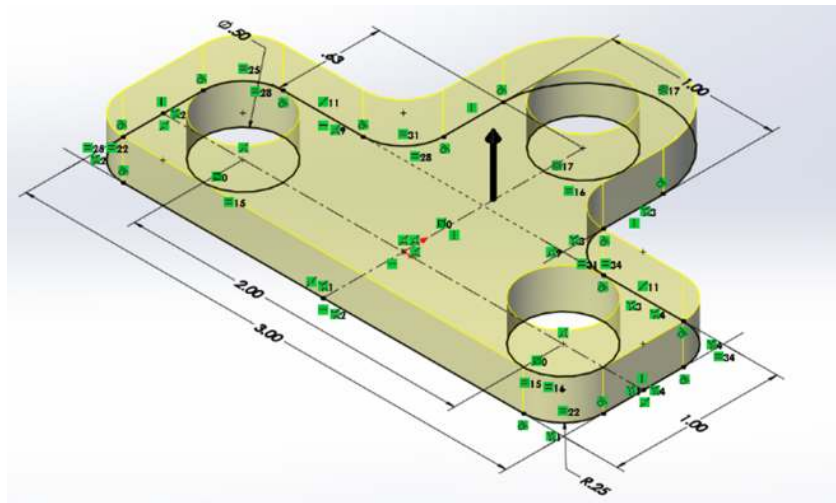


Figure 2.33 Extrude Preview

SKETCHING AND RELATIONS REVIEW

? Sketching and Relations Practice Drawings and Tutorials

Below are three drawing examples with the drawing PDF's for you to use to model the parts shown and tutorial videos to help you model the parts shown. Watch the videos one by one trying to model the part from the drawing after each video. If you need to, feel free to follow along with the video step by step.

- [2130002A PDF](#)
- [2130010 PDF](#)
- [2130203 PDF](#)





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CHAPTER OVERVIEW

3: Advanced Sketching Tutorials

Learning Objectives

By the end of this chapter, you will be able to...

- Create advanced sketches using sketch tools and relations.

Throughout this chapter of the text, we will complete a few complex-sketch part models in SolidWorks to grow our skills in working with sketches, using sketch creation and modification tools, and using geometric relations to fully define sketches.

[3.1: Bracket Tutorial](#)

[3.2: Oil Pan Gasket Tutorial](#)

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3.1: Bracket Tutorial

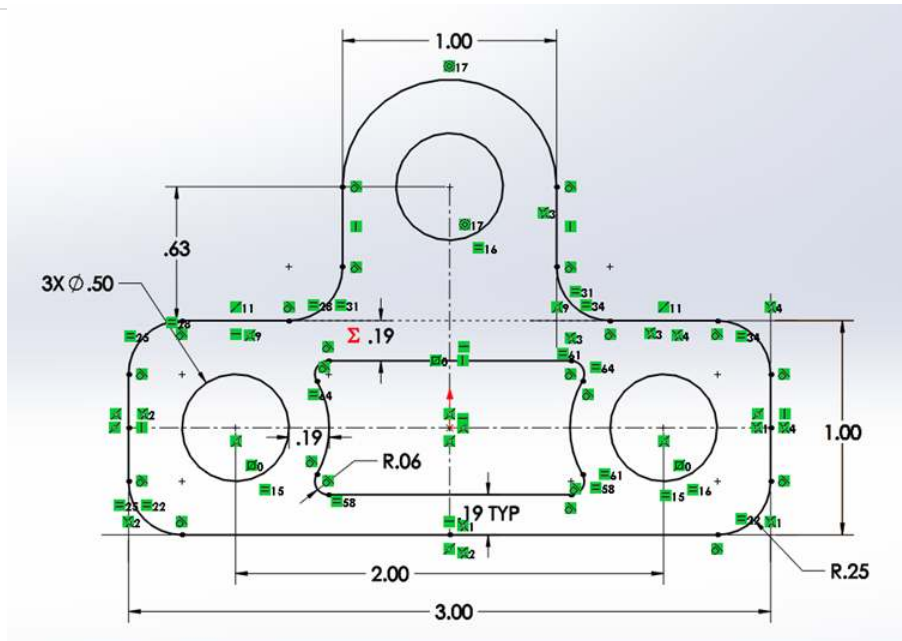


Figure 3.1 Completed Sketch with Fillets of 2130003

The part we will be creating in this tutorial is like the bracket shown in the previous chapter. You can find a PDF drawing for the part named [2130003 Drawing](#) linked.

First start SolidWorks and open a new part. If you get the default settings dialog box, choose IPS and ANSI for the units and drafting standard. Start a new sketch on the top plane by clicking the top plane in the FeatureManager Design Tree and selecting the “Sketch” icon.

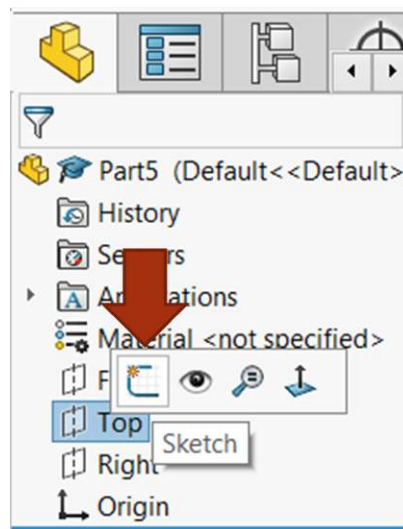


Figure 3.2 Start a New Sketch on the Top Plane

THE CENTER RECTANGLE

Activate the center rectangle command from the rectangle drop down on the Sketch tab. Before placing the first point, be sure to check the radial button in the property manager for “From Midpoints” (Figure 3.3).

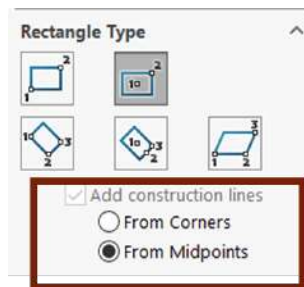


Figure 3.3 Ensure that the Radial Option From Midpoints is Selected

Then place the center point of the rectangle at the red origin, bring your cursor outward to expand the rectangle, click to secure the rectangle into place, and add Smart Dimensions on the length and width of the rectangle 3 inches and 1 inch.

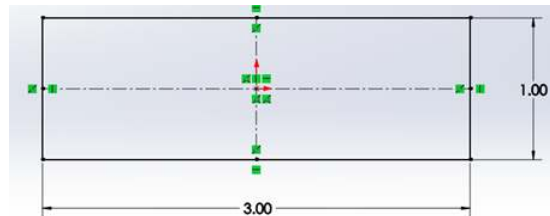


Figure 3.4 Three Inch by One Inch Rectangle

Draw two circles being sure to place their center points on the horizontal centerline, on the left and right sides (Figure 3.5). This will add automatic coincident relations. To do this, activate the circle command from the CommandManager, left click the horizontal center line on the left side of the vertical centerline and then bring your cursor outward to expand the circle, click to secure the circle into place, and do the same thing for the right side of the vertical center line. Add a Smart dimension to only the left circle at a diameter of .50”.

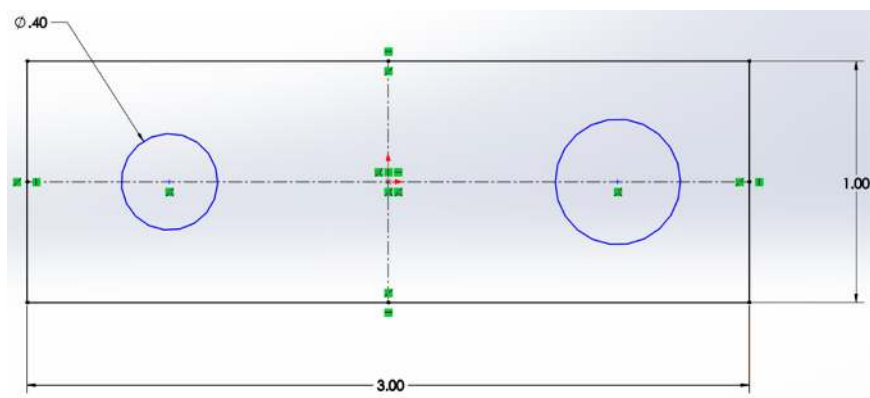


Figure 3.5 Draw Two Circles on the Horizontal Centerline and Dimension Only One

Activate the “Add Relation” command from the “Display/Delete Relations” drop down (figure 3.6) and then select the two circles along with the vertical centerline. When all three are selected, “Symmetric” should be an option for available relations: select the “Symmetric” relation (Figure 3.7). Also add a Smart dimension between the two circle centers that is two inches (Figure 3.8).

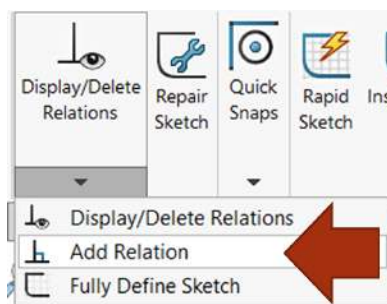


Figure 3.6 Activate the Add Relation Command

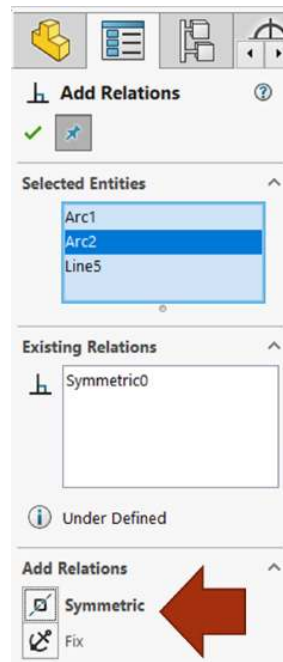


Figure 3.7 Make the Two Circles Symmetric Across the Vertical Centerline

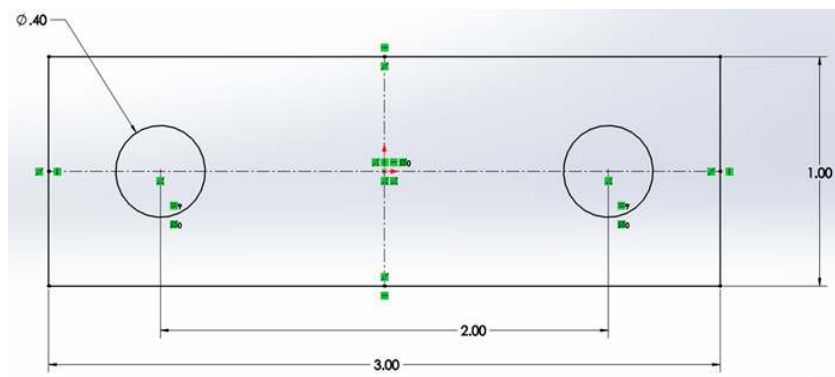
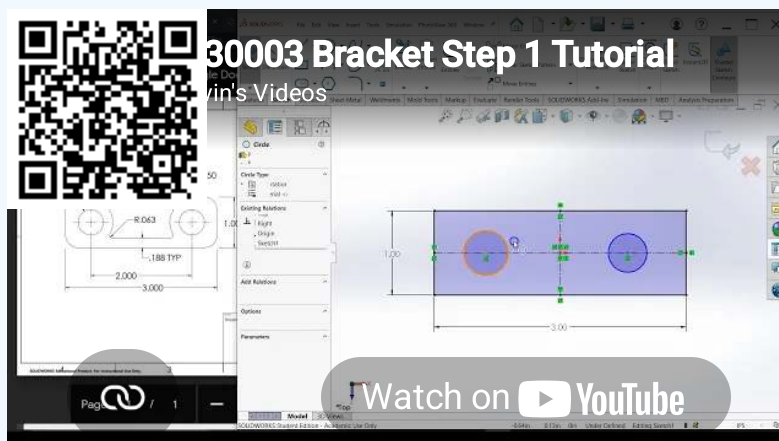


Figure 3.8 Two Circles Symmetric and Dimensioned

? Exercise 3.1: Video Tutorial of Center Rectangle Portion of the Bracket

Here is a video tutorial version of the instructions written above.



OFFSETTING AND TRIMMING A CUTOUT

After you have added the dimension, activate the Offset Entities tool (this will automatically get you out of the Smart Dimension tool). Offset the two circles outward .1875" as shown in figure 3.9. Click the “Reverse” box in the property manager if needed. Confirm the offset for the two circles.

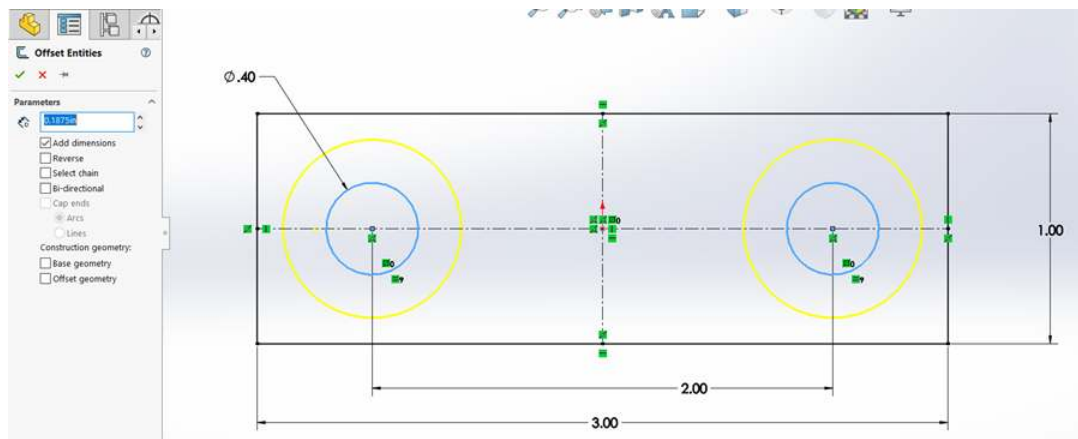


Figure 3.9 Offset Two Circles

Now offset the top and bottom line of the rectangle inward by the same amount (make sure “Select Chain” is not selected in the property manager and click one edge of the rectangle). You will have to activate the Offset Entities tool again. Confirm the command with the green check mark.

Note that you can left click drag the end points of the horizontal lines to make trimming easier in the next steps.

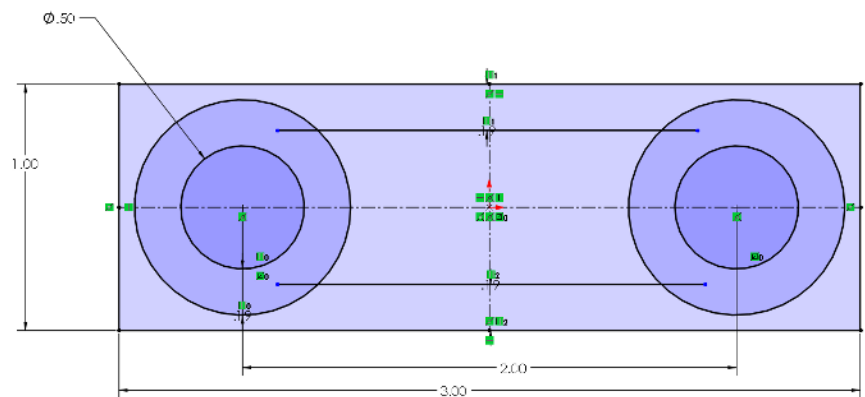


Figure 3.10 Offset Top and Bottom Lines of Rectangle

With all these lines everywhere, we must do some trimming to get closer to the cutout shown in the drawing. Activate the Trim Entities tool from the CommandManager. Left click drag through all the unwanted lines; your result should be that shown in figure 3.11. Confirm the trim tool with the green check mark when you are finished.

If you mess up the trim, hit the Undo command and try again.

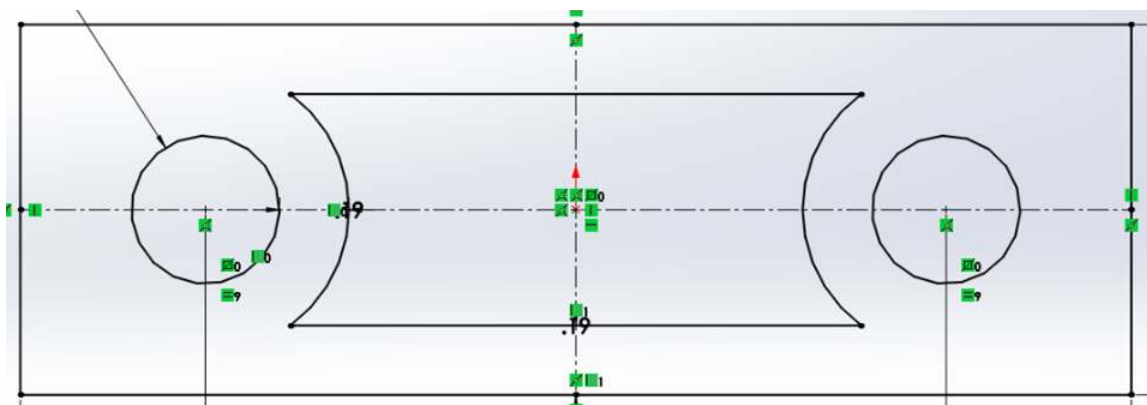
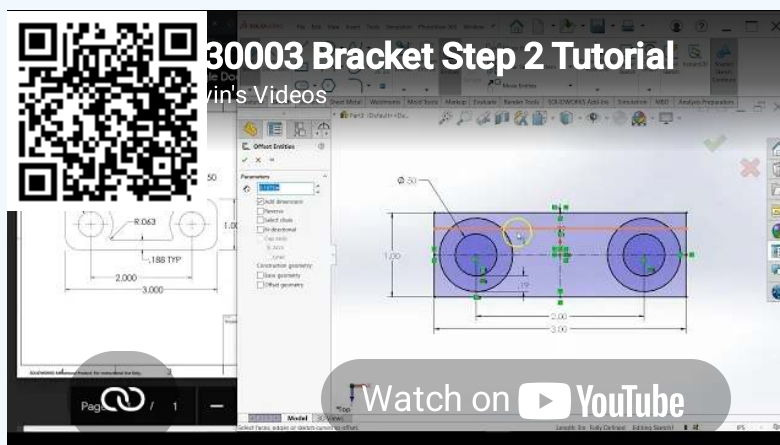


Figure 3.11 Trimming the Excess Offset Lines

? Exercise 3.2: Video Tutorial of Trimming and Offsetting the Cutout

Here is a video tutorial version of the instructions written above.



THE UPPER CLEVIS PORTION OF THE BRACKET

Activate the line command, draw a line starting from the left side of the top line of the rectangle upward, following the blue guide-lines for making an automatic vertical relation (Figure 3.12).



Figure 3.12 Start Line For Clevis Portion

Left click to confirm the line upward, then move your cursor out from the point and then back to the same point. When you move your cursor out again, you will notice that your rubber band line has become a rubber band arc. Bring the arc to the right past the middle of the part to create a perfect semi-circle. The blue guides will help. Click to confirm the semi-circle, then move your cursor directly downward to the upper horizontal of the rectangle. You will notice your line reverted back to a rubber band line. Click on the upper horizontal to confirm the endpoint of your line and double click fast to exit the line command. Confirm that your sketch matches figure 3.13.

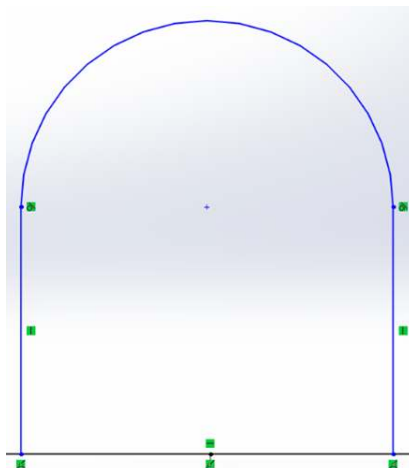


Figure 3.13 Full Clevis Portion Before Trim

Trim the portion of the upper horizontal line on the rectangle that is within your new arch sketch. Then left click drag the upper point of the vertical centerline in the middle of the part closer to the center of the arc. Activate the “Add Relation” command, select the center point of the arc so that both the top centerline point and center of the arc appear in the selection box of the “Add Relation” property manager, and select the “Merge Points” option (Figure 3.14).

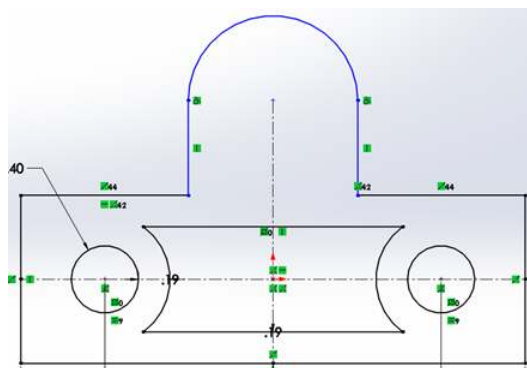


Figure 3.14 Full Clevis After Trim

Add a circle to the center of the arc (Figure 3.15). Rather than dimension it, add an equal relation to it between the other two circles. Then add a 0.625” dimension between the center of the arc/circle and one of the upper lines on the rectangle (click the circle on its perimeter and then click the line; click to place the dimension and enter “0.625”). Add a dimension between the left and right vertical lines. Place the dimension slightly above the sketch and then specify 1 inch for the dimension.

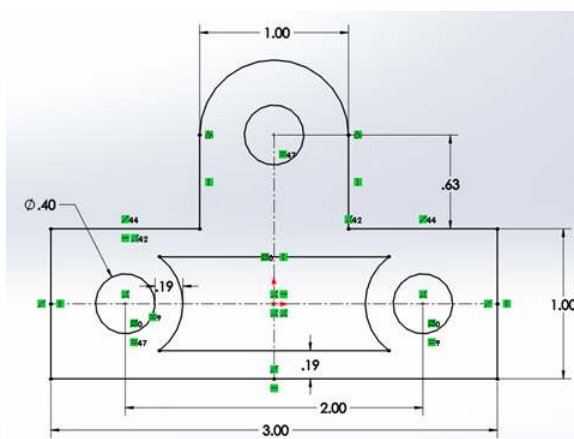


Figure 3.15 Merge Centerline to Clevis Joint

? Exercise 3.3: Video Tutorial of Upper Clevis Joint Sketch

Here is a video tutorial version of the instructions written above.



FILLET THE CORNERS AND EXTRUDE

Fillet all the outside corners to have a radius of .25" and fillet the inside corners to have a radius of .0625". Activate the Sketch fillet command, specify the radius of .25" in the size box. Click all 6 of the outside corners one by one. Hit the confirm green check mark for the fillet, change the radius in the size box to .0625" and click the 4 inside corners.

The final sketch is shown below again as it was at the beginning of this section.

Note: You may have to re-add the overall rectangle dimensions based on how they were first added. The outer fillets may break the dimensional constraints on the rectangular portion of the sketch. You can add smart dimensions again using the left and right edges of the rectangle and the top and bottom edges of the rectangle.

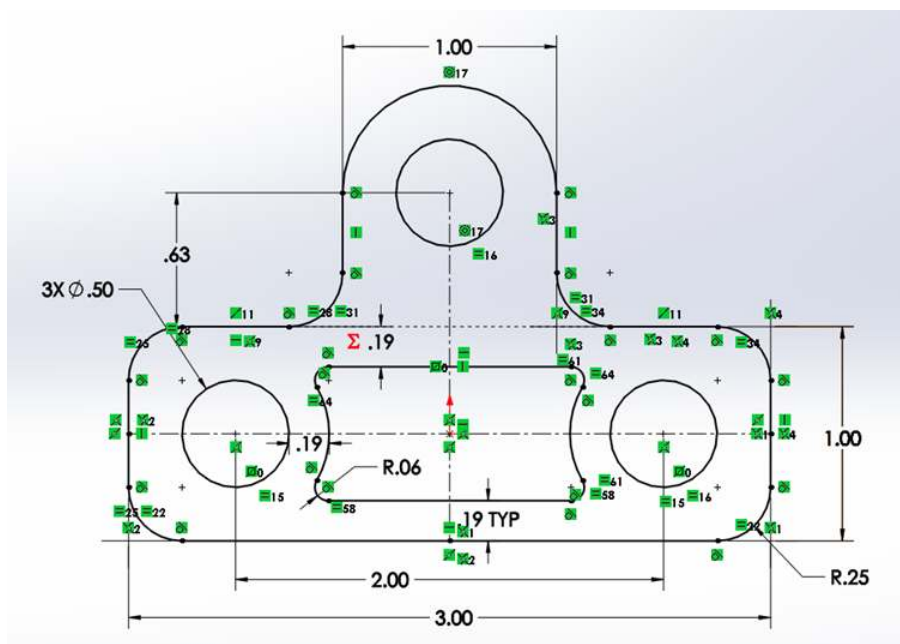


Figure 3.1 Completed Sketch with Fillets of 2130003

Activate the Extruded Boss/Base command by switching to the Features tab of the CommandManager (the command is on the very left-hand side). Set the depth to .25" in the property manager as is shown in figure 2.49 and confirm with a green check mark.

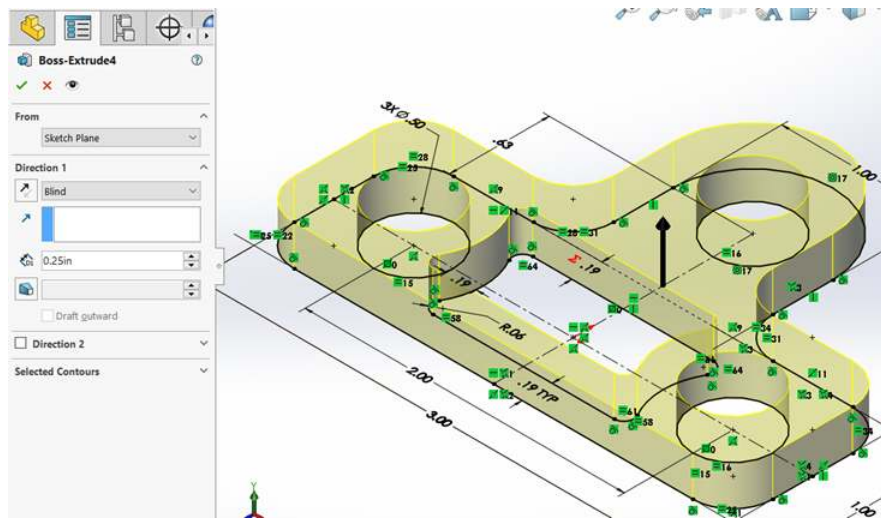
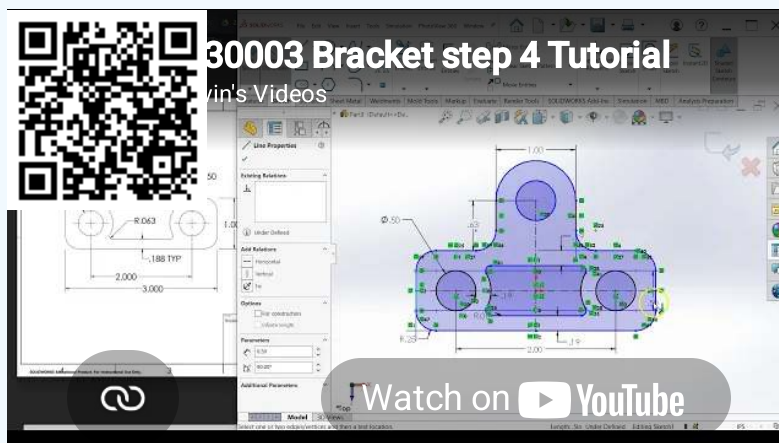


Figure 3.16 Extrude of 2130003

Save your part as **2130003**. You have finished this tutorial! Here is the [2130003 Answer Key](#) to check your work.

? Exercise 3.4: Video Tutorial of Fillets and Extrude

Here is a video tutorial version of the instructions written above.



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3.2: Oil Pan Gasket Tutorial

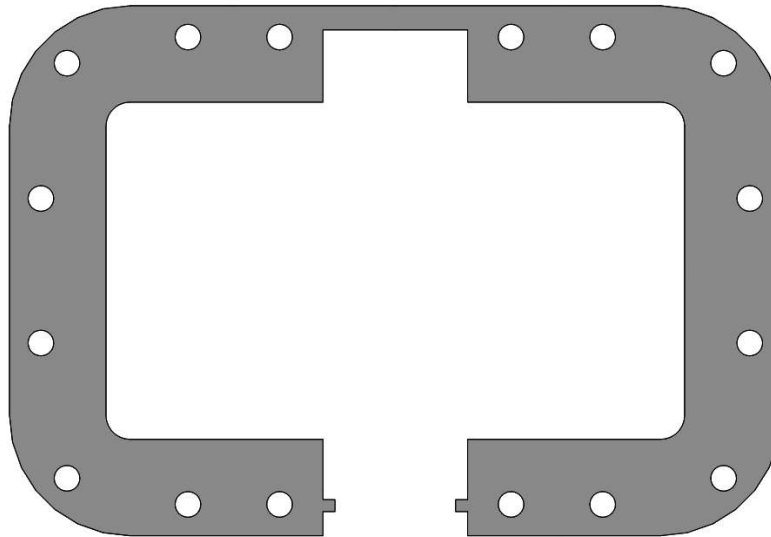


Figure 3.17 Oil Pan Gasket Top View

Here let's step up our sketching practice a bit with a gasket part that requires a significant amount of relations and modification tools. We will demonstrate the sketching of [2130205 Oil Pan Gasket](#) drawing linked.

THE CENTER RECTANGLE AND OFFSETS

First start a SolidWorks new part with ANSI Drafting Standard and IPS Units and start a new sketch on the **Top Plane**. Draw a center rectangle at the origin using the From Midpoints option that is 8 inches in length and 5.5 inches in width (Figure 3.18).

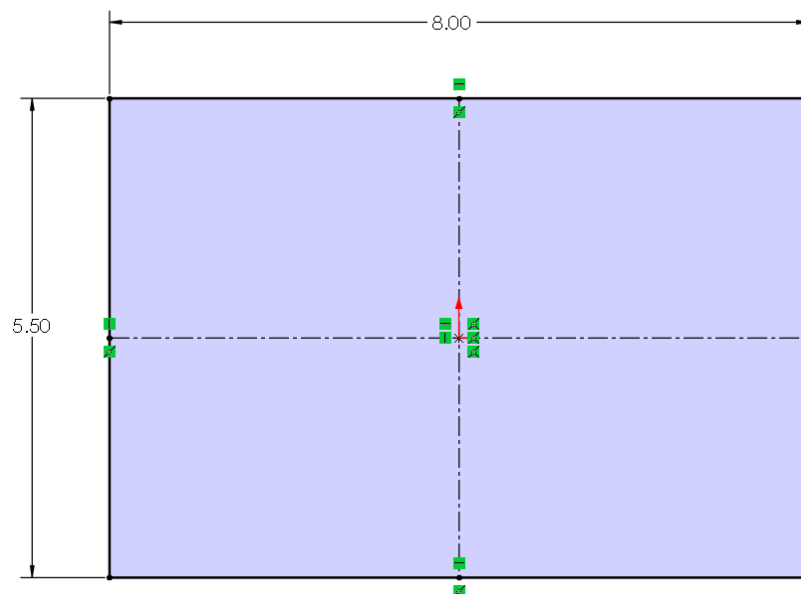


Figure 3.18 First Center Rectangle

After adding the center rectangle and dimensioning it using the Smart Dimension, use the fillet sketch tool to add 1.25 inch fillets to the four corners of the center rectangle. We are adding the fillets this early into the sketching process because we want to offset the outer contour inward with the fillets using the Offset Entities tool.

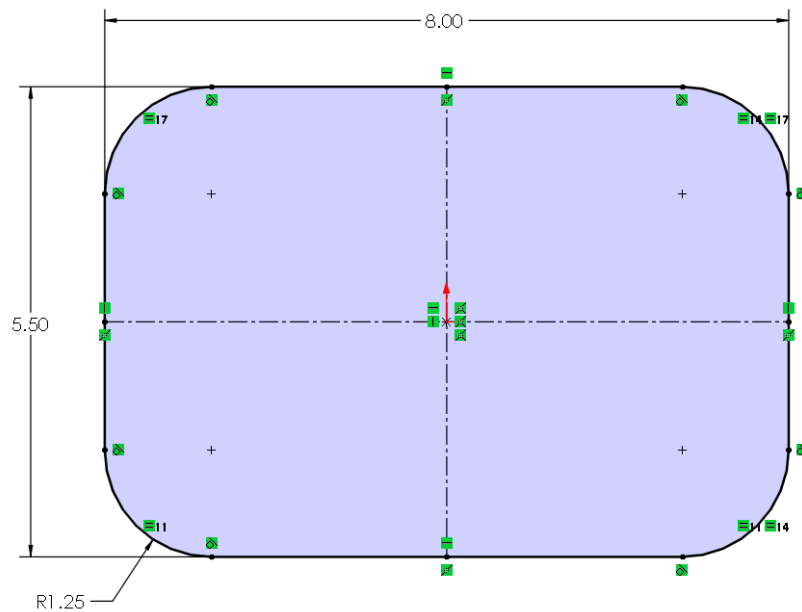


Figure 3.19 First Center Rectangle with Fillets

Using Offset Entities, offset the rectangle inward by 1 inch making sure that **Select Chain** is checked. Click on one of the edges of the outer contour and with Select Chain enabled, the offset entities tool will collect all the connected edges of the contour. Note that a 1 inch dimension is automatically added.

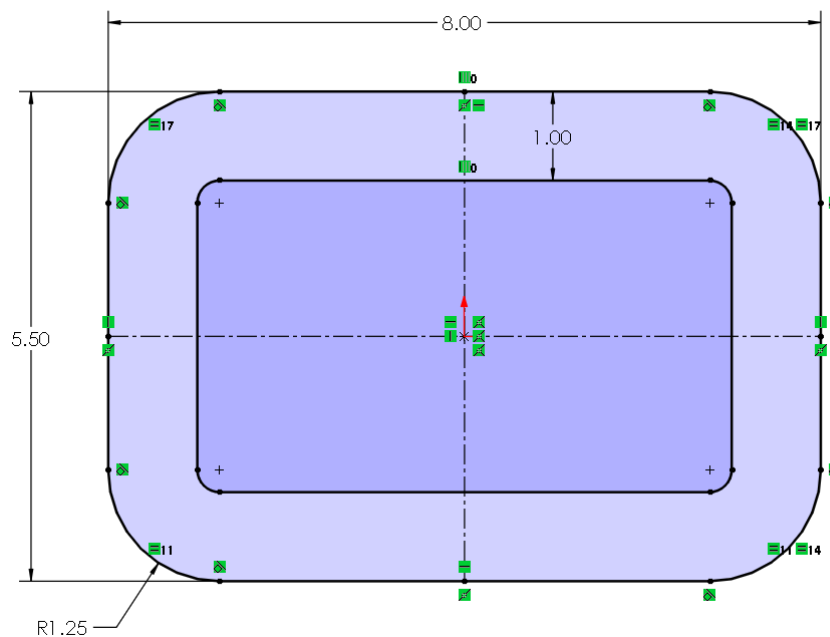


Figure 3.2.1: Copy and Paste Caption here. (Copyright; author via source)

To get the length and width of the inner rectangle to show in a future drawing, you can add those dimensions to your sketch by making those dimensions driven. You will get a dialog box that says you are over defining the drawing but it will give you the option to either make the dimensions driven or to leave the sketch over defined.

Use Offset Entities again to offset the outer contour by .325 inches as construction geometry. To make the offset geometry construction geometry, under **Construction Geometry** in the property manager, activate the **Offset Geometry** option.

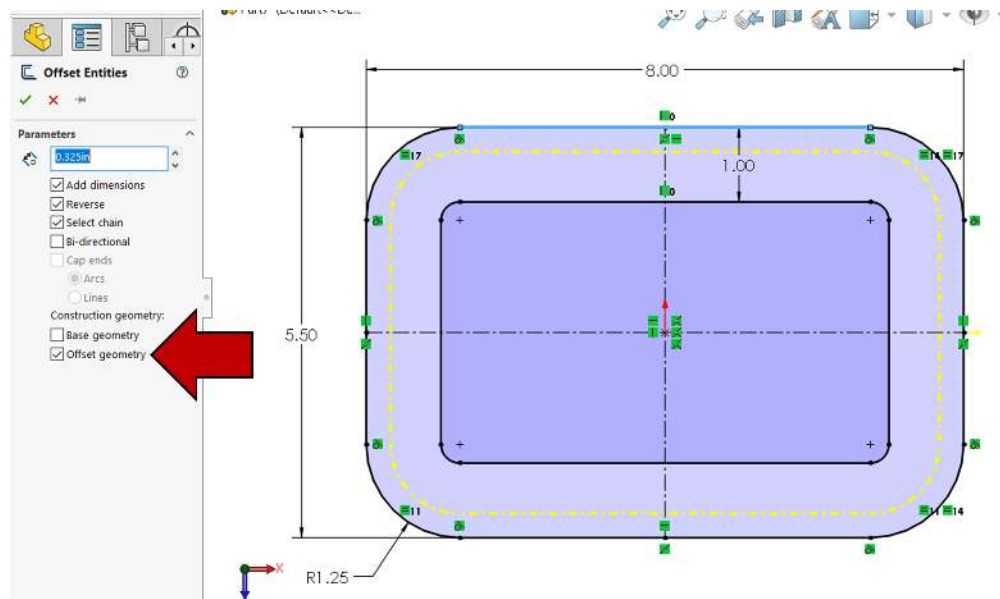


Figure 3.21 .325 Inch Offset Construction Geometry

? Exercise 3.5: Video Tutorial of Filleted and Offset Rectangles

Here is a video tutorial version of the instructions written above.



CREATING THE CUTOUTS

Using the Line tool, draw a rough sketch of the top middle cutout making sure to add the automatic coincident relations to the inner contour (Figure 3.22).

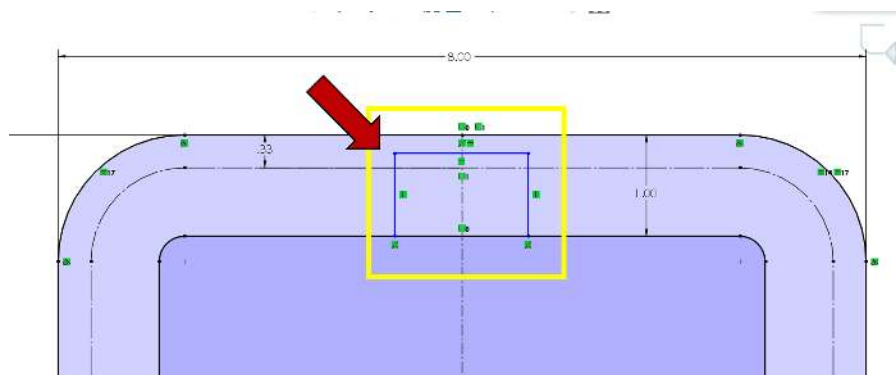


Figure 3.22 Top Cut Out Rough Sketch

Trim away the excess lines making sure not to trim the vertical centerline. Fully define the cutout with a 1.50 inch length, .25 inch thickness from top edge, and by making the left and right edge of the cutout symmetric with the vertical centerline (Figure 3.23).

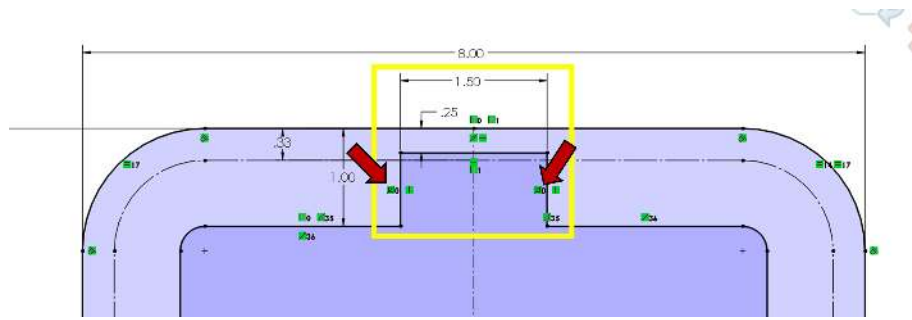


Figure 3.23 Top Cutout Fully Defined

Draw a rough sketch of the bottom left side of the cutout (don't do the right side because we will mirror it over). Make sure that your end points automatically have coincident relations with the inner and outer contour (Figure 3.24).

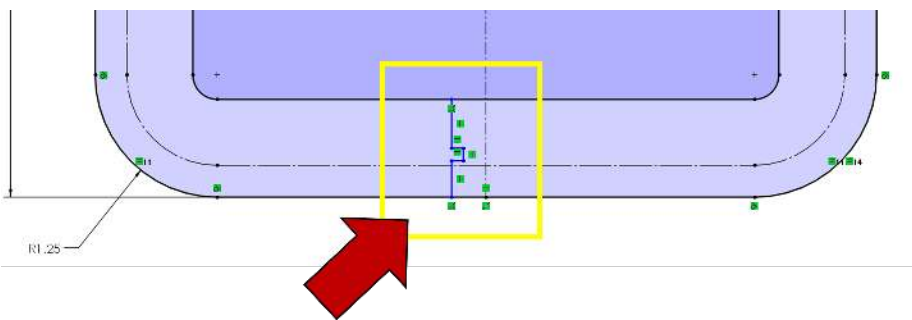


Figure 3.24 Rough Sketch of Bottom Left Side Cutout)

Mirror the left side to the right side using the vertical center line and then trim the excess lines away making sure not to trim away the vertical centerline (Figure 3.25). You will also need to make the endpoint of the vertical centerline coincident with the bottom outer contour edge or draw a construction line connecting the outer contour at the bottom and then making the vertical centerline endpoint midpoint related to the new construction line.

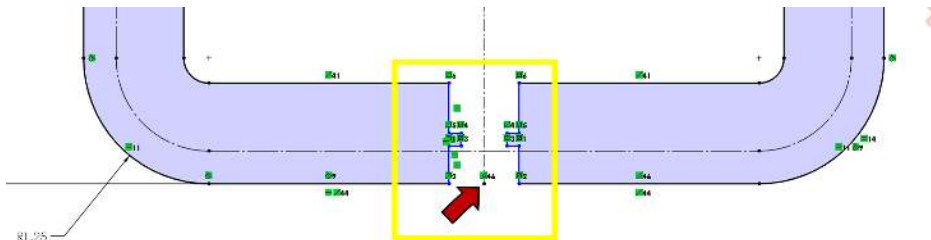


Figure 3.25 Mirror and Trim of Bottom Cutout

After making sure that the trim was successful and adding the coincident relation was successful, fully define the bottom cutout with a 1.5 inch spacing, 1.25 inch spacing on the tabs, a .25 inch distance from bottom edge on the outer contour to the bottom edge of one of the tabs, a tab thickness of .125 inches and a colinear relation between the left aligned lines of the cutout (Figure 3.26).

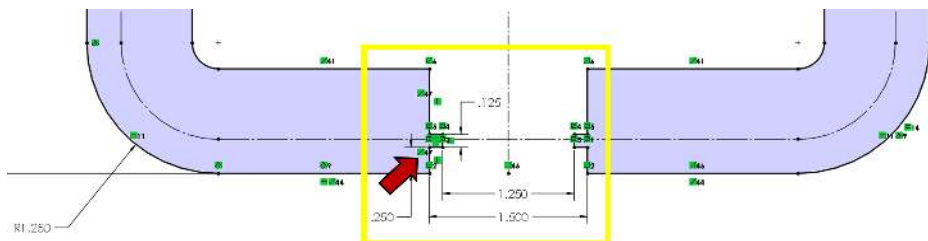


Figure 3.26 Fully Defined Bottom Cutout

? Exercise 3.6: Video Tutorial of Creating the Cutouts

Here is a video tutorial version of the instructions written above.



CLEARANCE HOLE PLACEMENT

The clearance holes in the Oil Pan Gasket are symmetrical across both the vertical centerline and the horizontal centerline. So, we only need to create and fully define the holes in one quarter of the part and then we will mirror the holes across the vertical and horizontal centerlines.

Using the Circle tool, place and fully define four .266 inch diameter holes as shown in figure 3.27. The right most hole is dimensioned from the vertical centerline by 1.200 inches, the second to right most hole is dimensioned from the vertical centerline by 2.15 inches. Make the four holes equal by CTRL selecting all four holes and choosing the equal relation. The hole at the fillet corner is coincident with the midpoint of the fillet on the construction offset; this can be added automatically by snapping to the construction arc's midpoint. The left most hole is .75 inches from the horizontal centerline.

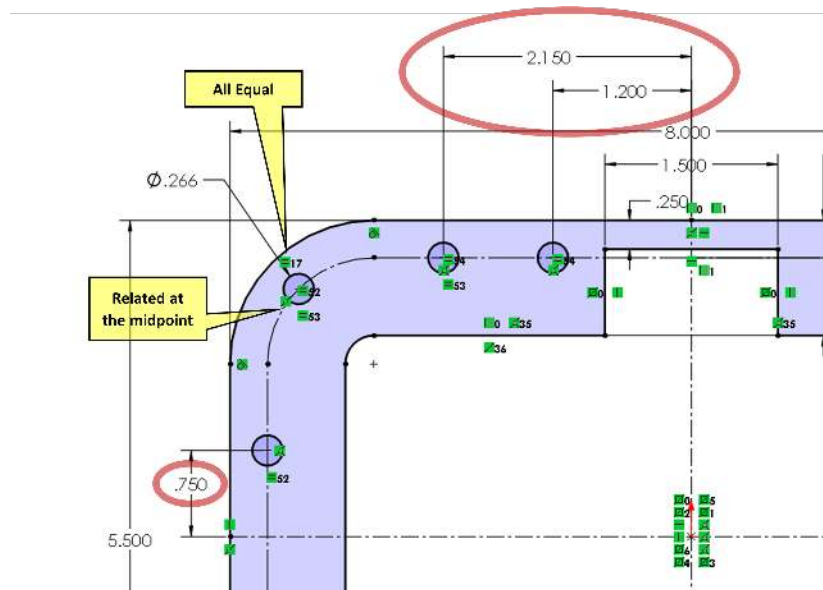


Figure 3.27 Definition of Clearance Holes

When clearance holes are fully defined in the top left corner of the sketch, use **Mirror Entities** to first mirror all the clearance holes across the vertical centerline and then use Mirror Entities a second time to mirror all eight holes across the horizontal centerline. You will see a lot of symmetry relations crowd the drawing now (Figure 3.27).

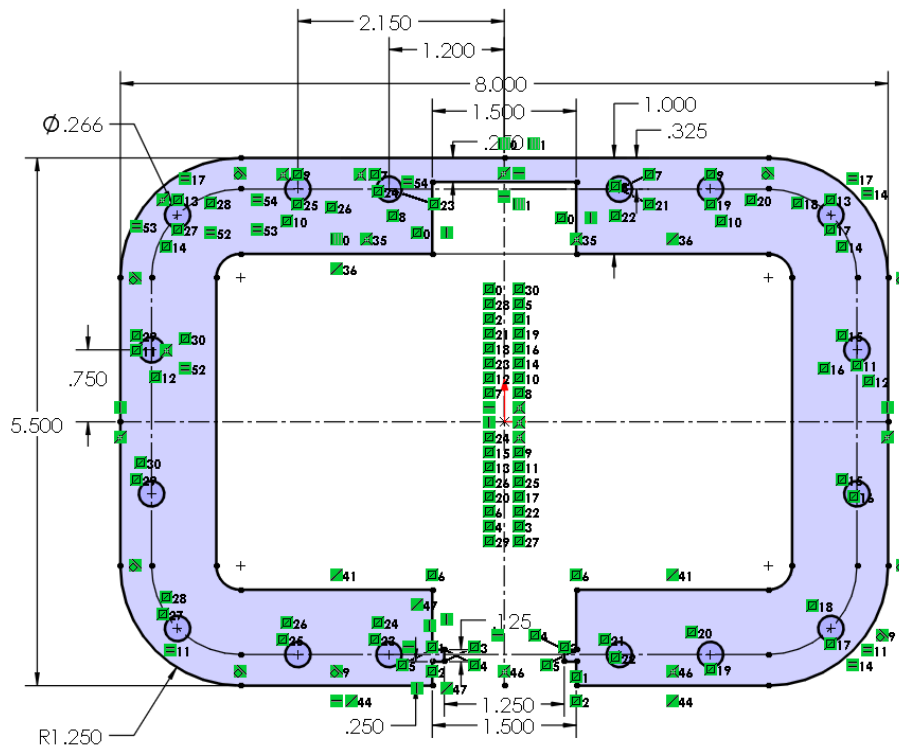


Figure 3.28 Clearance Holes Mirrored

Use the Extruded Boss/Base feature to extrude the sketch .005 inches and save as **2130205**.

? Exercise 3.7: Video Tutorial of Clearance Holes and Extrude

Here is a video tutorial version of the instructions written above.



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CHAPTER OVERVIEW

4: Extruded Features

Learning Objectives

The purpose of this chapter is going to be to...

- Use the expanded settings of Boss Extrude.
- Use Cut Extrude, Hole Wizard, Patterning Features, Fillet, Chamfer.
- Create multi-feature parts.
- Begin evaluating models for maximum efficiency.

In the previous chapter, we explored creating single sketch parts that took a single sketch and extruded the sketch using the Extruded Boss/Base feature. In this chapter, we will explore features in more depth, begin modeling multi-feature parts, explore feature-based features, and features that remove material.

[4.1: Sketch-Based Features](#)

[4.2: Feature Based Features](#)

[4.3: CAE1001 Full Part Tutorial](#)

[4.4: CAE1002 Full Part Tutorial](#)

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4.1: Sketch-Based Features

EXTRUDED BOSS/BASE

We had exposure to Extruded Boss/Base in the last chapter but didn't explore its full potential. The Extruded Boss/Base feature has many options controlling the depth of a feature: how deep the feature is extruded, which direction is the feature extruded, does the extrude taper, does it go up to a certain point or is offset from a certain point, etc.

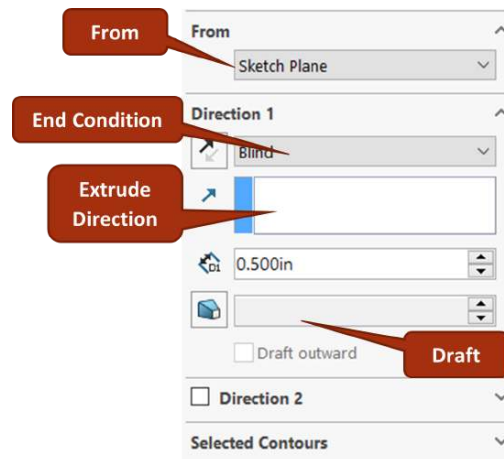


Figure 4.1 Extruded Boss/Base Property Manager

When you start a boss extrude, either from a sketch or with intent to specify a new sketch, you are able to modify the property manager for the extrude feature. The following options are available:

FROM

Specify where your extruded feature will begin. The most common option is from the sketch plane, but you might want to start it from a different face/plane/vertex or offset it from the sketch plane for special situations.

END CONDITION

Specify where extruded feature will end.

- **Blind:** You can have it end a distance from the sketch plane
- **Up to Vertex:** The feature will extrude to the depth of a selected point.
- **Up to Surface:** The feature will extrude to the depth of a selected face or plane.
- **Offset from Surface:** The feature will extrude to the depth of a selected face or plane plus/minus a specified offset distance.
- **Mid Plane:** A popular option that allows the extruded feature to be centered about the sketch plane and end an equal distance from the sketch plane on both sides.

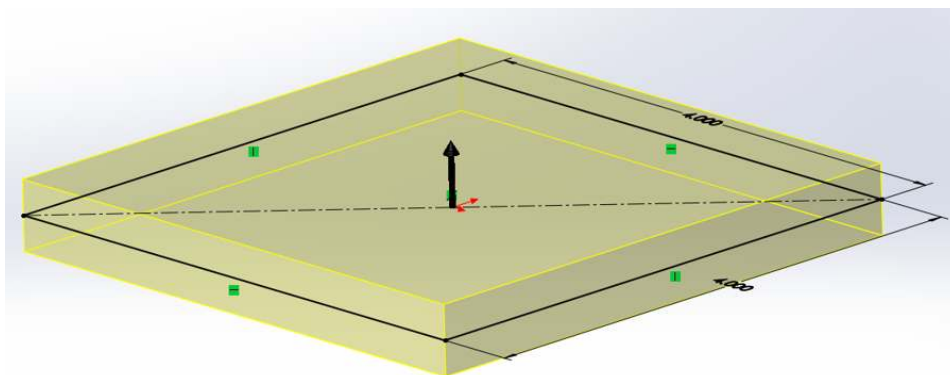


Figure 4.2 Example of Midplane Extrude

EXTRUDE DIRECTION

The default direction is 90 degrees from the sketch plane so you don't usually need to specify anything in this box, but for instances when you want to extrude at an angle different from 90, you can select a model edge or reference line to specify a new direction.

DRAFT

Draft adds a taper to the extruded feature along the depth of the feature.

DIRECTION 2

You can specify a second direction usually going in the opposing direction of the first. This is useful for specifying different depths/conditions from the sketch plane (which becomes the center of the extrude in this scenario).

SELECTED CONTOURS

If you don't want to extrude your entire sketch, you can specify the areas (or contours) that you want to extrude. This is a great efficiency tool we will learn about later that when used can reduce sketch time by almost a factor of ten. It eliminates the need to trim which is one of the most time-consuming tasks in sketching!

? Exercise 4.1 Extruded Boss/Base Options

Let's practice manipulating a few of the Extruded Boss/Base options to get familiar with what each does and how we can use them to model towards design intent. With the model we are going to begin creating, be sure to save it and keep it open as CAE1001 because we will continue to use it in following exercises to complete a multi-feature part.

- Open a new part in SolidWorks with IPS units and your preferred drafting standard.
- Create a simple sketch on any of the default planes in a new SolidWorks part.
- Extrude the sketch and practice changing various parameters in each of the sections described above.



EXTRUDED CUT

The **Extruded Cut** tool is another extrude feature that is used to make cut outs, pockets, etc. into your existing solid model.

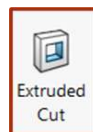


Figure 4.3 Extrude Cut Button

An extruded cut can't be your first feature in a part because you must cut into something, so you at least need to have one extruded feature in the model before you can use cut extrude. Extruded cut is a Sketch-Based Feature, so you start a sketch on a plane or a face in the model, sketch your cut profile, and then use cut extrude to specify the details of the depth of cut.

The options and method for using the extruded cut tool are much the same as the Extruded Boss/Base Tool. You have the same From, End Condition, Direction, Draft, and Selected Contours Options.

SKETCHING ON MODEL FACES

Sketching on a model face is very similar to sketching on a plane. The key advantage however is that when you sketch on a model face, your extruded features will by default originate from that face. This fact is very useful in making the part shown in figure 4.4.

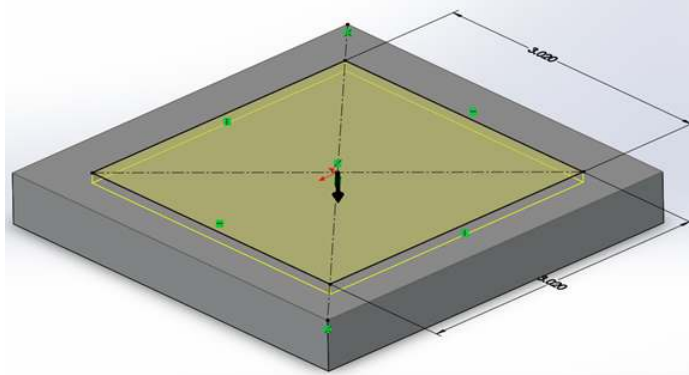


Figure 4.3 Using Cut Extrude from a Sketch on Model Face

When sketching in a model that already has a feature in it, in your new sketch you can utilize the edges and faces from that already existing feature. For example, say for the model in figure 4.4 instead of specifying a length and width for the rectangular cutout, you wanted to specify that the rectangular cutout be an offset distance away from the exterior rectangle. You can then select the face of the exterior rectangle and use the offset command to offset the internal rectangle the distance you desire. You can also use **Convert Entities** to reuse model edges in a new sketch.

CUTTING WITH OPEN SKETCHES

Another option that exists with the Cut Extrude tool is cutting models with open sketches. For example, if you wanted to saw off tubing at a 45-degree angle, instead of drawing a closed sketch that would completely miss your design intent, you can just draw a line at 45 degrees (Figure 4.5).

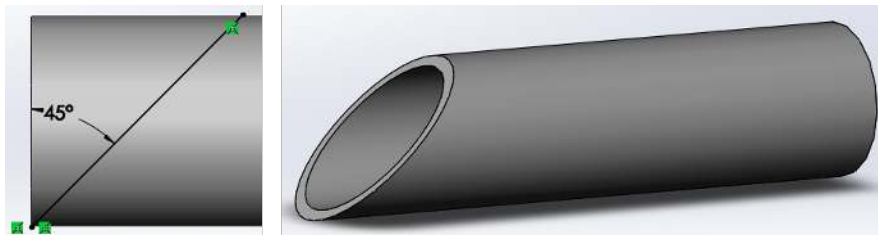


Figure 4.5 Using an Open Sketch to Cut

? Exercise 4.2: Using Cut Extrude in SolidWorks

Using the same model from the previous exercise, CAE1001, we will create a cut-extrude on the top face of the part and explore the different cut extrude options.

- From the CAE1001 model, start a sketch on the top rectangular face.
- Draw a 3.020" by 3.020" center rectangle centered on the origin.
- Extrude a blind cut .100" into the material.
- Explore other options with the cut extrude and with using existing model edges in a sketch on a model face.



HOLE WIZARD

The **hole wizard** is a macro tool that combines sketching, extrudes, and revolved features into one highly customizable tool for creating standard holes. Think of the holes that fasteners and pins go into; this is the tool to quickly specify these kinds of holes! The hole wizard has two tabs: The type tab and position tab (Figure 4.7).

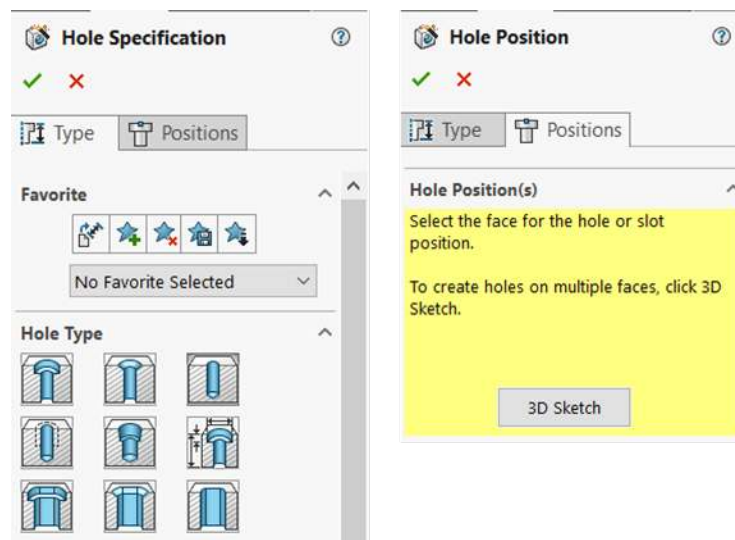


Figure 4.7 Hole Wizard Property Manager

TYPE TAB

The **type tab** is used to specify the type of hole you want to specify such as whether it is a counterbore, countersink, tapped hole, etc. This is also where you specify the depth of your hole or if your hole goes all the way through the part, if it has a nearside or far side countersink, if the hole bottom needs to be flat or can have the shape of a drill bit, and much more.

POSITIONS TAB

The **positions tab** is for placing your hole. This tab works more as a sketch where you select the face (this can be a planar or a curved face) where your hole will start and then sketch to define the location. Holes are placed with the point sketch tool. You have access to all the sketch tools you usually have in a sketch when on the positions tab of the Hole Wizard.

? Exercise 4.3 Using Hole Wizard Holes

Here we will experiment with the various hole types available within the Hole wizard tool. Most standardized hole types used in mechanical design are available within the hole wizard tool.

- With CAE1001, place a hole on the bottom center of the part thus far by activating the Hole Wizard command, switching to the positions tab, selecting the bottom face, and then placing a hole at the center.

- Note that holes are placed in Hole Wizard using the **point** sketch tool on the sketch tab of the CommandManager.
- To best get familiar with the Hole Wizard tool, experiment with changing different settings such as the hole type, end condition, check boxes, etc. on the Type tab.



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4.2: Feature Based Features

It has been previously stated that the above features are called Sketch-Based Features. In SolidWorks, feature-based features open many new possibilities for model shapes and increased efficiency in modeling. Feature-Based Features make modifications to already existing features based on common design parameters of the mechanical designer.

Some examples of feature-based features are...

- Pattern Features (Mirror, Linear Pattern, Circular Pattern, etc.)
- Fillet and Chamfer
- DFMTTools (Shell, Draft, Dome, Rib, etc.)

PATTERN FEATURES

These features such as mirror, linear pattern, and circular pattern should already look familiar to you from chapter 2 of this text. These options exist in the sketch environment, but there is much more power in applying these types of features in the feature environment than in the sketch environment.

MIRROR

The mirror feature can be used to mirror one or more features across a mirror plane. Sometimes you must create the mirror plane but most of the time, the front, top, or right default plane will do. When you activate the mirror property manager (figure 4.8), you select the features you want to mirror and the mirror plane.

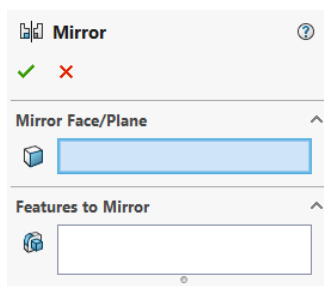


Figure 4.8 Mirror Property Manager

In the example shown below in figure 4.9, the top cut out is being mirrored onto the bottom of part CAE1002 by using the top plane as the mirror plane. For this to work, the main boss extrude needs to be centered about the top plane on creation.

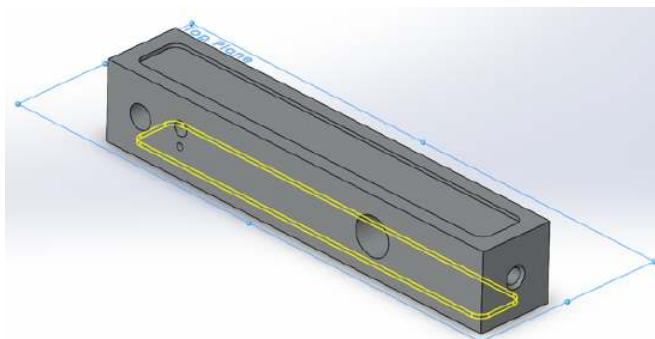


Figure 4.9 Example Use of Mirror Feature

CIRCULAR PATTERN

Circular pattern is one of the more popular and common features due to the occurrence of patterns in cylindrical parts such as hubs, flywheels, gears, etc. The circular pattern tool is like the sketch pattern tool except instead of selecting a point as the pattern axis, you select a reference axis or feature representing an axis to specify an axis for the pattern. This is because we are working in 3D and to truly define a circular pattern in 3D, we must center the pattern with a 3-dimensional object such as an axis.

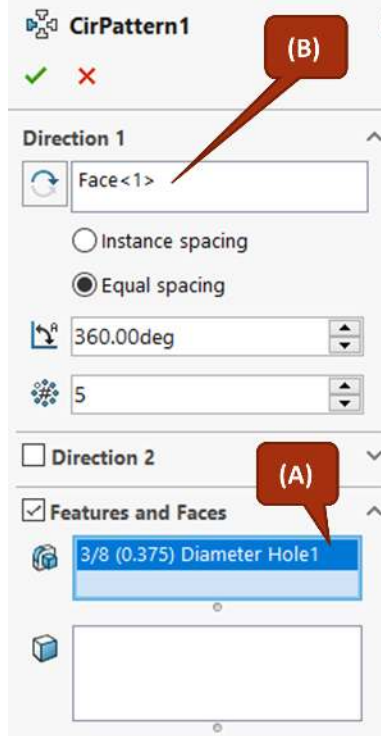


Figure 4.10 Circular Pattern Property Manager

In the example shown in figure 4.11 using circular pattern, when the circular pattern tool is activated, the top 3/8 hole is selected as the feature to be patterned (A) and the center hole in the center of the part is then selected as the reference for the pattern axis (B).

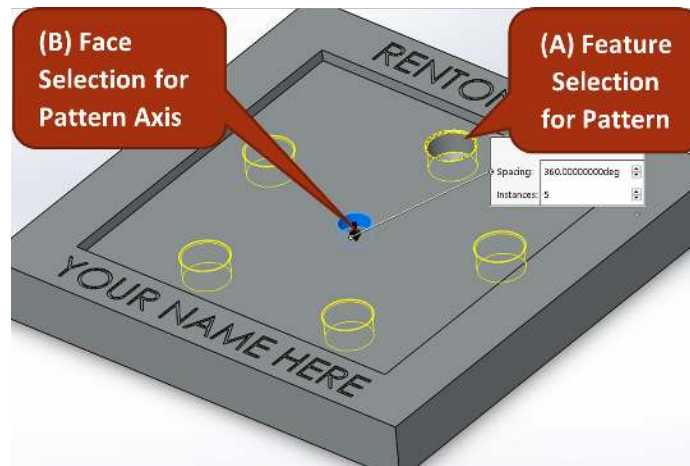
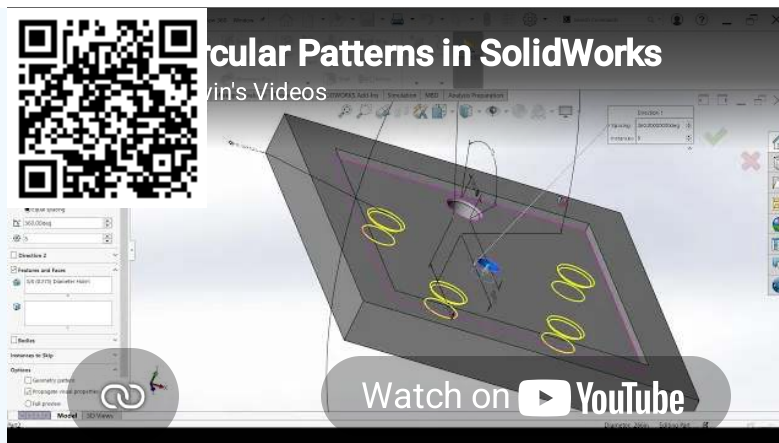


Figure 4.11 Example Use of Circular Pattern Feature

? Exercise 4.4: Practice Circular Pattern

Let's make a circular pattern from the 3/8" hole on the on the bottom face of the pocket in CAE1001. It's as simple as selecting the geometry to be patterned with a reference axis such as the center counterbore hole.

- First activate the circular pattern tool from the Linear Pattern drop down on the Features tab of the CommandManager.
- Then select for features to pattern the top 3/8" hole, also specifying equal spacing and how many holes will be created for the pattern.
- Then select the inner face of the counterbore hole as shown above.
- Specify the spacing and number of instances to be in the pattern.



LINEAR PATTERN

Linear pattern works much of the same way as circular pattern. However, instead of selecting a pattern axis, you select pattern directions. You can select either one or two pattern directions. You also have the option of specifying either a number of copies and spacing for the pattern or patterning up to a reference with specified spacing from the reference edge.

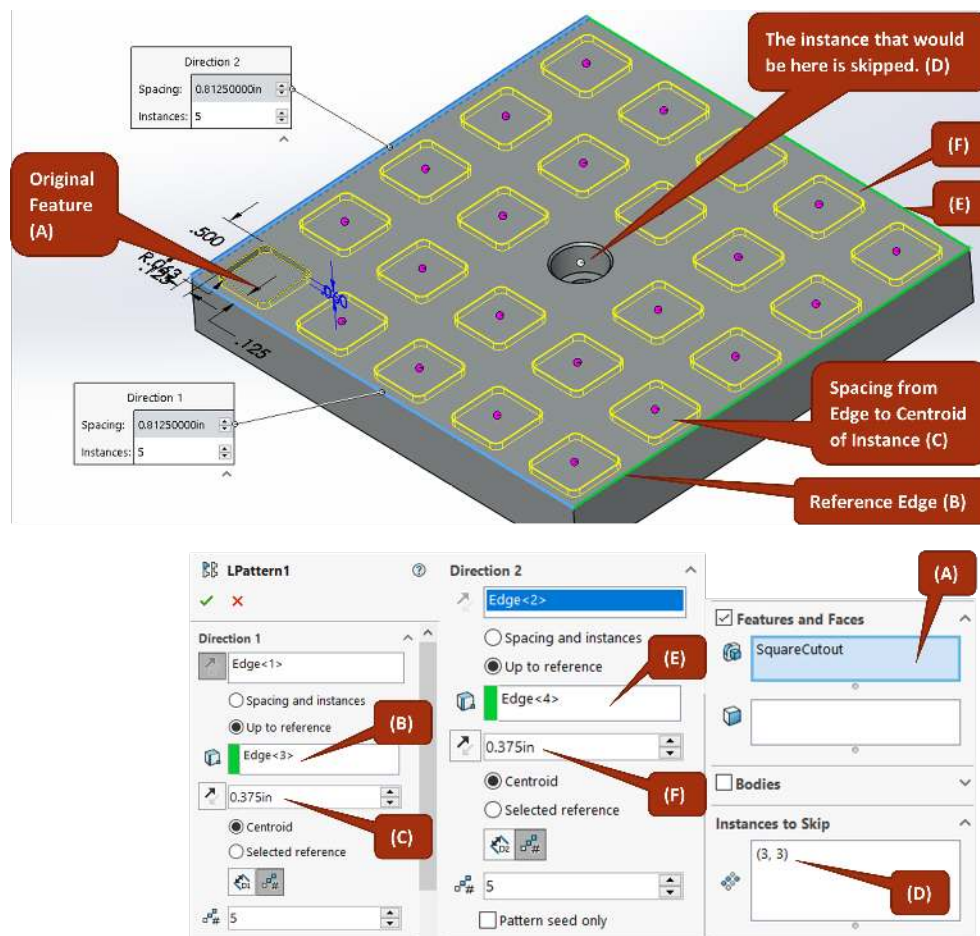


Figure 4.12 Example of Linear Pattern with Reference Edge

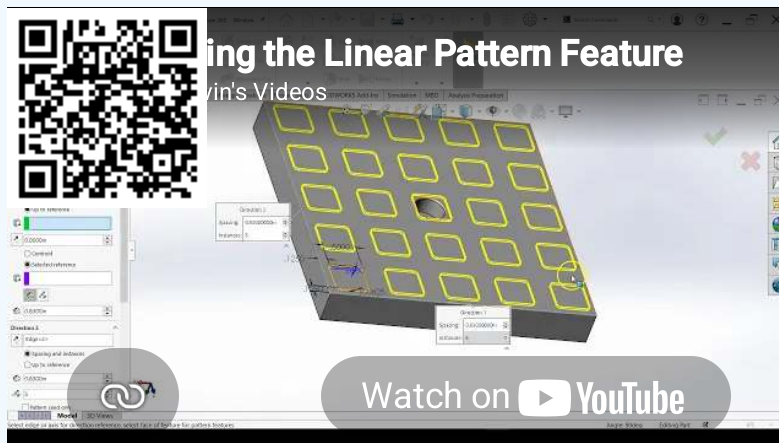
In the example shown in figure 4.12, a 0.5-inch square in the bottom left edge of the part is patterned in the x and y direction. The feature is patterned up to a reference using the opposing edge of the part as the reference for both the x and y. Measurements specified in the property dialog box are from the centroid of the feature being patterned, so .375 inches is specified in the reference offset to give .125 inches of space from the reference edge (same as the originating feature).

The number of instances is then set to five for both directions and the middle instance is skipped to avoid interfering with the counterbore hole that was already there. In this example, because the feature is being patterned to a reference, the spacing between each individual instance is automatically calculated and doesn't need to be specified.

? Exercise 4.5: Using Linear Pattern Feature

Let's explore all the options available with the linear pattern feature that enable you to specify your design intent. We will see how to use linear pattern with set spacing and instance count, how to pattern up to a reference, spacing from a reference edge on the original instance, and skipping instances.

- Sketch and extrude a cut of a square cutout with fillets built into the sketch on the bottom face of CAE1001.
- Activate the linear pattern feature, select your feature to patter, reference edges for direction 1 and 2, and choose up to reference.
- Choose the opposing edges as the reference edges for direction 1 and 2.
- Choose to space based off a reference edge and choose the reference faces applicable on the original square cutout.



FILLET AND CHAMFER

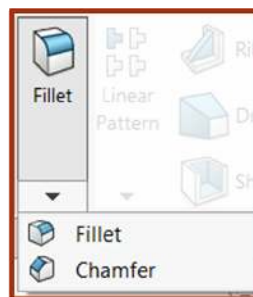
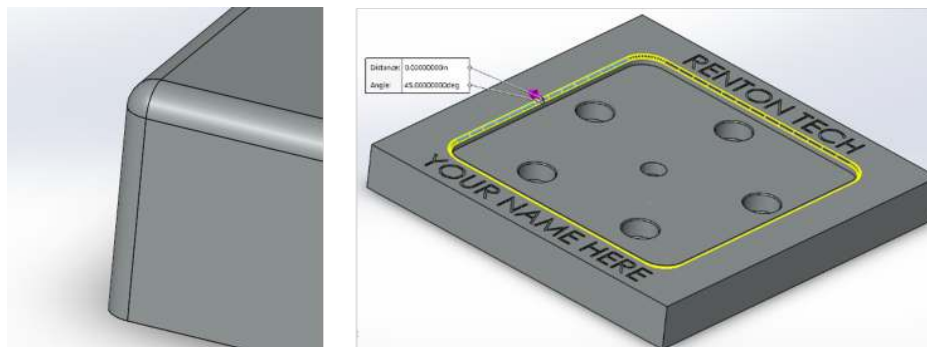


Figure 4.13 Fillet and Chamfer Features

The fillet and chamfer tool are very straightforward and intuitive: you specify the radius or chamfer size you want and then click **edges**. With fillet and chamfer, you can select edges and faces to create 3D fillets that can't be done in Sketch fillet or sketch chamfer. In addition, all the fillets and chamfers that are possible with sketch fillet and chamfer can be done using the feature fillet and chamfer.

Many designers will prefer to do all fillets and chamfers using the feature fillet and chamfer tool. The fillet and chamfer feature tools will allow you to customize your fillets and chamfers with extreme specificity.



Fillet specifically has a smart context toolbar that allows you to select model elements faster. When you select a singular edge for fillet, the context toolbar appears with options for other edges to automatically select. You can select as many fillets or chamfer of the same size as you want in just one fillet or chamfer feature. You only need to make a new fillet or chamfer feature when you want to change the size.



It is good design practice to save your hole wizard, pattern, fillet and chamfer features for as late in the modeling process as possible in the order stated.

Add the specified fillets and chamfers to CAE 1001 using the fillet and chamfer feature tools.



WORKING WITH MULTI-FEATURE PARTS

There are editing capabilities that are possible when working with multi-feature parts. Some of the main benefits of this are...

- Being able to reorder features in the FeatureManager Design Tree by left click dragging features.
- Being able to roll back to an earlier stage of the model and add more features at that stage using the Rollback Bar. Left click drag the Rollback bar to adjust model timeline.

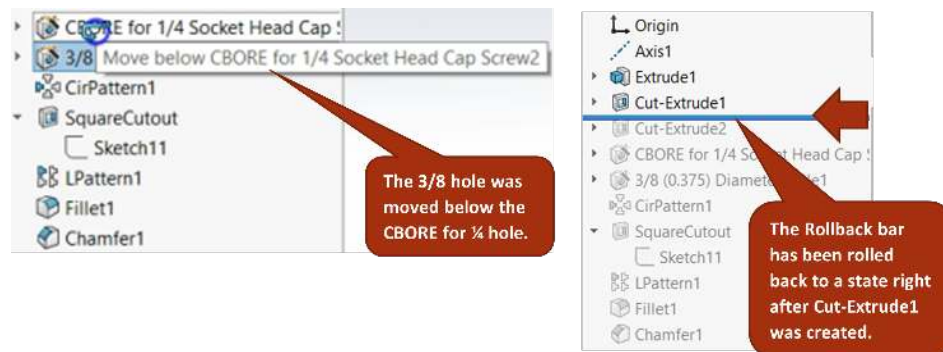


Figure 4.17 Feature Reorder and Rollback

These are important capabilities because many times in the design process, there are design changes that necessitate other features occurring earlier in the model or needing to add foundational features earlier in the model. Without these capabilities of reordering and rolling back using the Rollback bar, you would have to redo all the features that come after the point you need to change.

? Exercise 4.7: Rollback and Feature Reorder

One or two features should be reordered in CAE1001. Reorder them so that best design practice is followed with extrudes and base features at the beginning, hole wizard features after, pattern next, and finally fillet and chamfer features at the end of the feature manager design tree. Also, rename each feature so that it has an easily recognizable name.



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4.3: CAE1001 Full Part Tutorial

To apply the tools, we learned in this chapter we will model CAE1001 which incorporates Cut Extrude, Hole Wizard, Circular Pattern, Linear Pattern, Fillet, and Chamfer. You can find the design specifications for the part in the [CAE1001 Drawing](#). This tutorial will also be accompanied with the CAE1001 Video Tutorial at the bottom of this page.

BASE RECTANGLE

Open SolidWorks and **start a new part** being sure that the units for the part are inches (IPS) and the drafting standard is the one you desire (ANSI or ISO). **Start a new sketch** on the **top plane** and sketch a **center rectangle** that is 4 inches by 4 inches. Feel free to use an equal relation to make the rectangle a square. **Extrude** the rectangle to half of an inch.

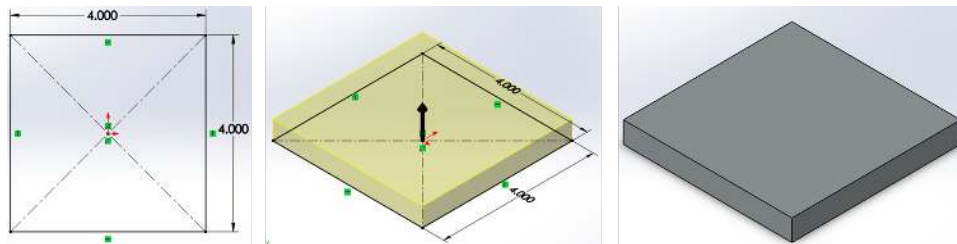


Figure 4.18 Extrude a 4x4x.5 rectangle

OFFSET POCKET

Start a sketch on the top face of the rectangle, select the top face of the part (it will highlight blue), then choose the Offset Entities tool. **Offset** the exterior edges of the face inward by 0.49 inches (you may have to check the reverse option). When finished, **select the Extruded Cut command** from the CommandManager and cut your sketch to a depth of 0.100 inches.

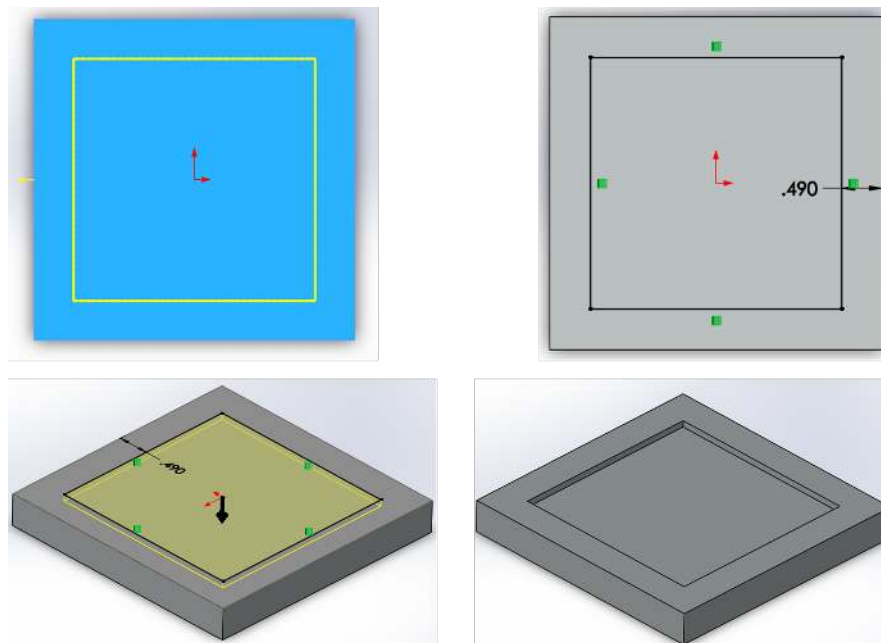


Figure 4.19 Create the offset pocket with cut extrude

ENGRAVE TEXT

Start another new sketch on the Top face of the part, **draw two centerlines** .120-inch (using smart dimensions) from the bottom edge and the top edge of the cut out making sure that the lines start and end on the edges of the part. **Activate the “Text” Sketch Tool**, Click the top centerline, specify Center justify in the property manager, type “RENTON TECH”, change the font to .250 inch text height and “Century Gothic” font (you will have to uncheck the “Use document font” box). **Repeat for the bottom centerline** but with your name. **Cut extrude** the sketch to a .010-inch depth.

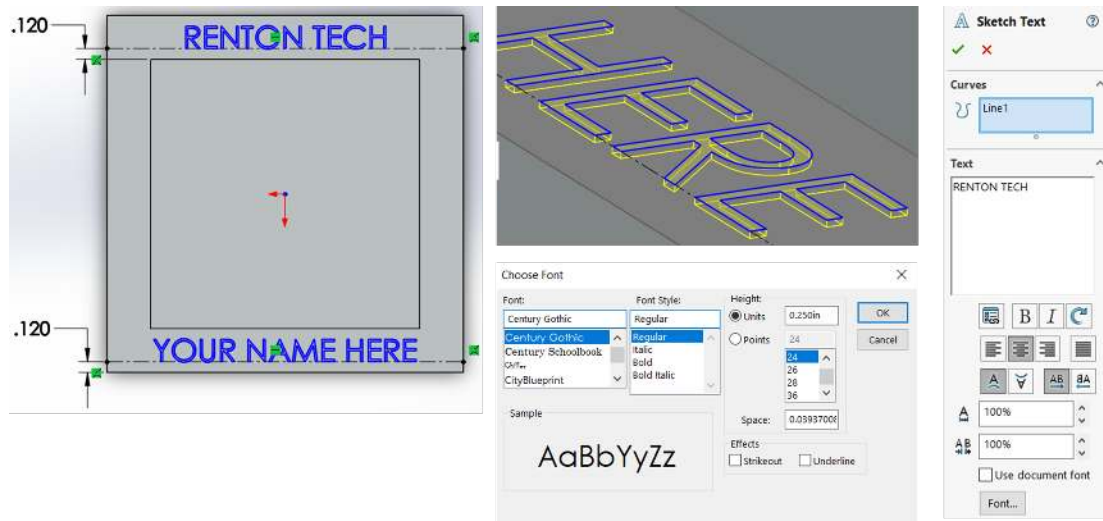


Figure 4.20 Extrude Engrave Text on the Top Face

HOLE WIZARD HOLES

Activate the **Hole Wizard** command and specify on the “**Type**” tab an ANSI Inch counterbore hole for a Socket Head Cap Screw that is a ¼ inch normal fit and goes through the entire part (Through All). Also specify a head clearance of .015 inches and a Near side countersink of .4750 inches at 90 degrees under the options section.

Flip your model over to the bottom side, switch to the **Positions** tab of the Hole Wizard Property Manager, select the bottom face of the part and then click the origin of the part. Confirm the hole wizard tool.

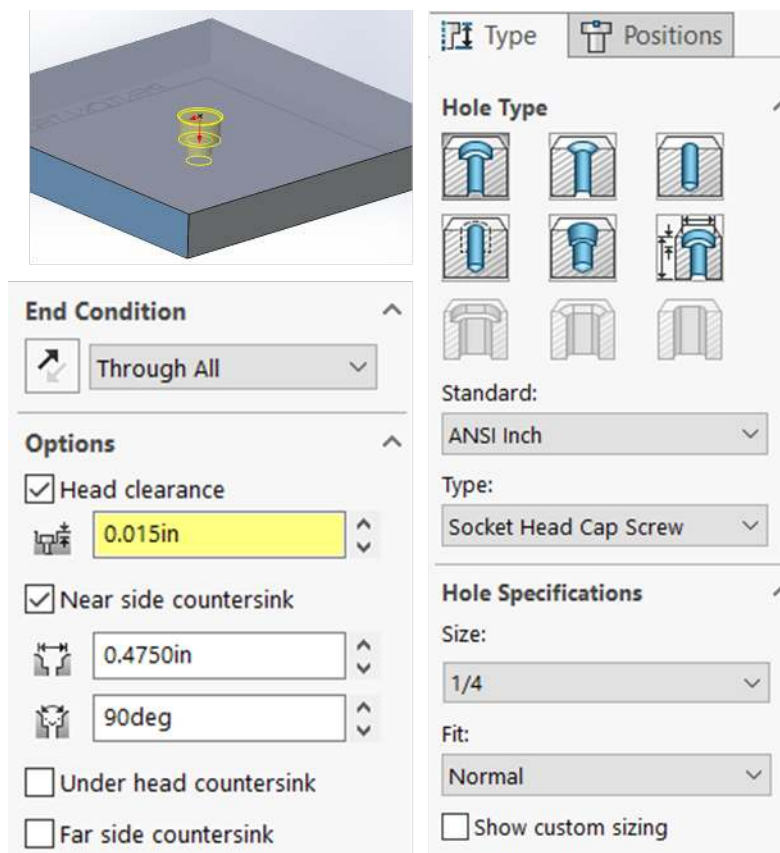


Figure 4.21 Create Counterbore Hole on Bottom Face

Activate the **Hole Wizard** again and specify on the “**Type**” tab an ANSI Inch Hole for a Fractional Drill Size of 3/8 of an inch and depth (End Condition) of .200 inches. Also, in the Options section of the property manager, **specify** a Near side countersink of .400 inches at 90 degrees.

Flip the model back over to the upright top side and switch the Hole Wizard to the **Positions** tab. **Unselect** the point tool from the CommandManager, **activate** the **circle** command and draw a circle from the origin. **Add** a smart dimension of 2.375 and switch the circle to be a construction circle. **Re-Activate** the sketch point command and **place the point** (which is the hole) at the Top quadrant of the circle. **Confirm** the Hole Wizard.

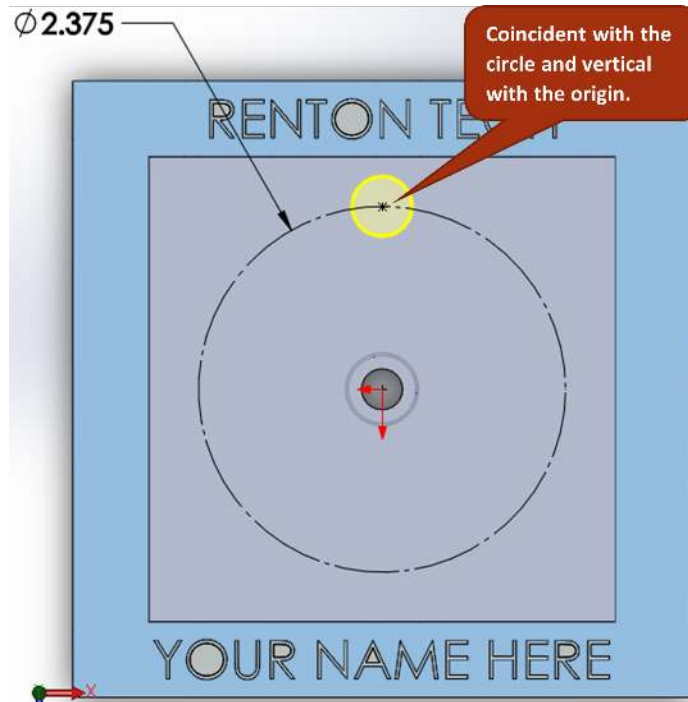


Figure 4.22 Add a Fractional Hole Within the Pocket

CIRCULAR PATTERN

Activate the **Circular Pattern** command from the Linear Pattern drop down. Select the 3/8 hole for features to pattern. Then, click into the top selection box of the property manager for specifying the axis of the pattern; select the center counterbore hole from the graphics area. Be sure to specify five for the number of instances and to check the equal spacing check box.

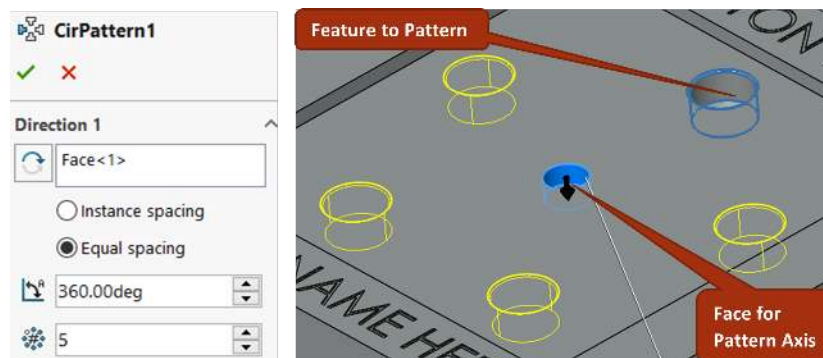


Figure 4.23 Circular Hole Pattern

SQUARE POCKET LINEAR PATTERN

For creating the linear pattern feature on the bottom side of the part, **flip the part over to the bottom**, **start a new sketch** and draw a .500-inch square that is .125 inches from the bottom and left edge. You will need to add Smart dimensions and potentially

relations. Also, **fillet the four corners** of the rectangle to have a radius of .0625 inches. Cut extrude the sketch .030 inches into the bottom of the base.

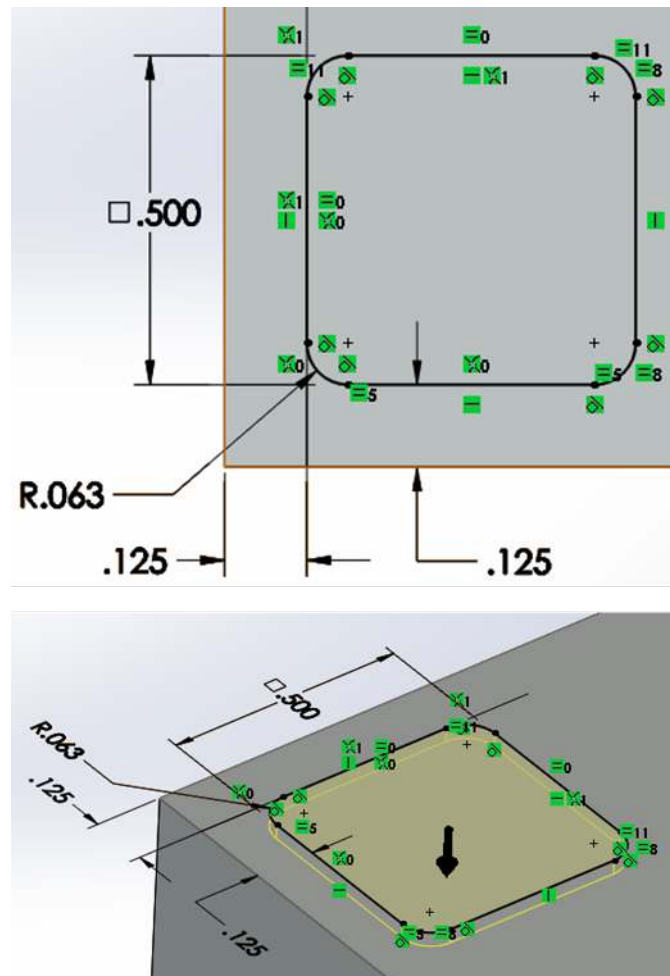


Figure 4.24 Square Pocket for Pattern

Activate the linear pattern feature from the CommandManager on the Features tab. **Click the horizontal bottom edge** to specify direction 1 and then **click the vertical left edge** to specify direction 2; then **click inside the square cutout** to specify the square cutout to be patterned.

Under the direction 1 and direction 2 settings in the property manager, **choose the “Up to Reference” option**, specify the opposing edges (right side edge for direction one and top edge for direction 2) in the selection box with green next to it. **Specify the offset distance** for both as .125”. **Choose “Selected Reference”** and choose the right and top face of the square pocket, switch the pattern option to “Set Number of Instances” and specify 5 for the number to pattern for both direction 1 and 2.

Also in the property manager near the bottom, **click inside the “Instances to Skip” box** (you may have to expand it first) then **click the very middle instance’s pink dot** to remove it from the pattern.

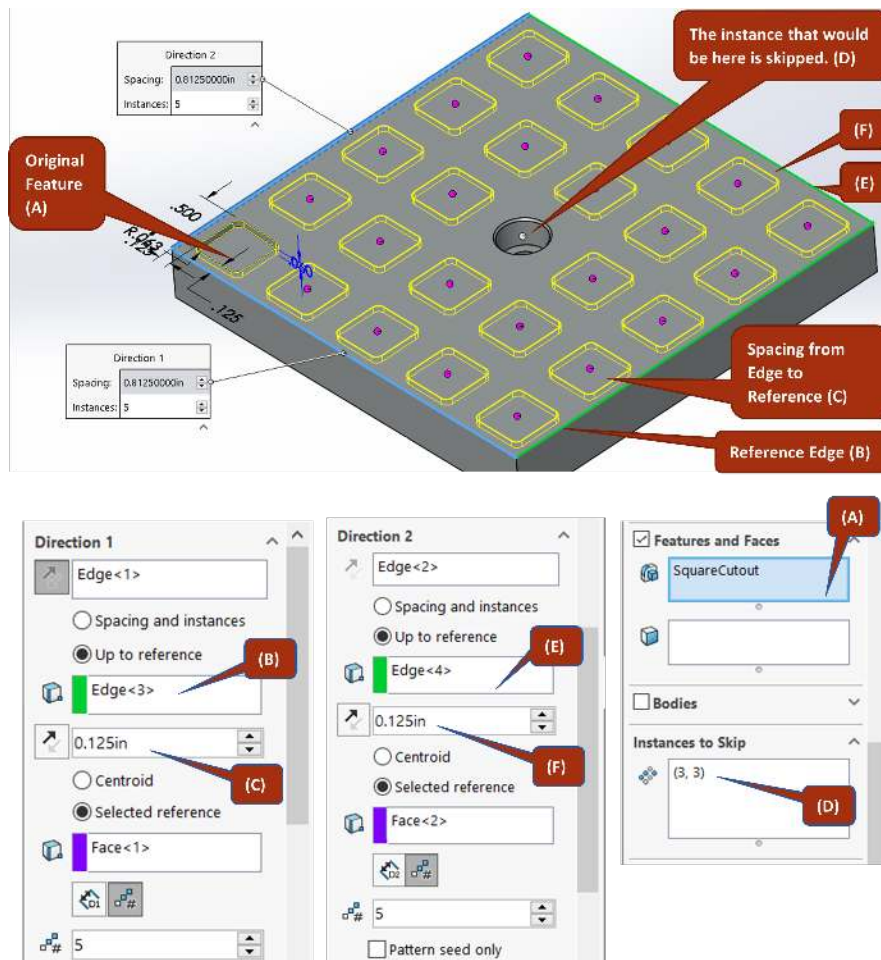


Figure 4.25 Linear Pattern to Reference

Before confirming the pattern, **experiment with the “Pattern Seed Only” option** which is right underneath Direction 2 settings to see the effect. We will leave it checked off for our tutorial, but it helps to know what that button does to the pattern.

FILLETS AND CHAMFERS

Next, **Activate the fillet command** from the CommandManager, set the fillet radius to $\frac{1}{4}$ inch, and select one of the inside corners of the top rectangular cutout. When the **context toolbar** appears, choose the first leftmost option to automatically select the other three corners at once. **Confirm** the command.

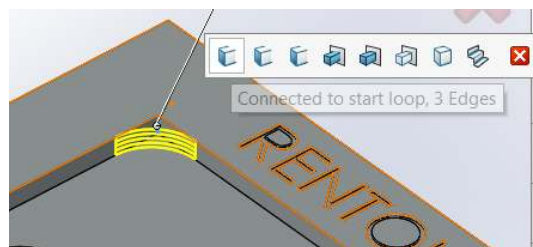


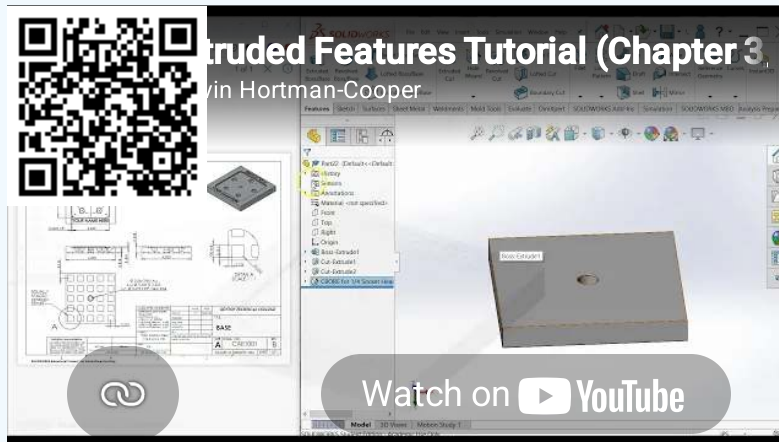
Figure 4.26 Fillet Corners

Activate the Chamfer command from the CommandManager, set the size to .030 inches and 45 degrees, and **select the top edge of the rectangular cutout**. Confirm the command.

Right Click on “Material <Not Specified>” in the FeatureManager Design Tree, select “Plain Carbon Steel” for the material, check your mass via “Mass Properties (should be 1.895 pounds), then **save your part!**

? CAE1001 Video Tutorial

If you are struggling to follow along with the written instructions, feel free to follow along with the video tutorial as well to make sure you are getting everything.



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4.4: CAE1002 Full Part Tutorial

Here let's do another tutorial practice to get more familiar with creating multi-feature parts. Being able to visualize the individual features of a complex part is critical to gaining proficiency in parametric modeling software as well as successfully passing the CSWA exam. Here we will model **CAE1002** based on the CAE1002 Part Drawing. CAE1002 is another part in the CAE1000 Compressed Motor Assembly that is modeled throughout this text.

BOSS AND CUT EXTRUDES

To begin, open a new part with IPS units and your preferred drafting standard. Sketch a one inch square on the Top Plane and extrude upwards to 5.660 inches.

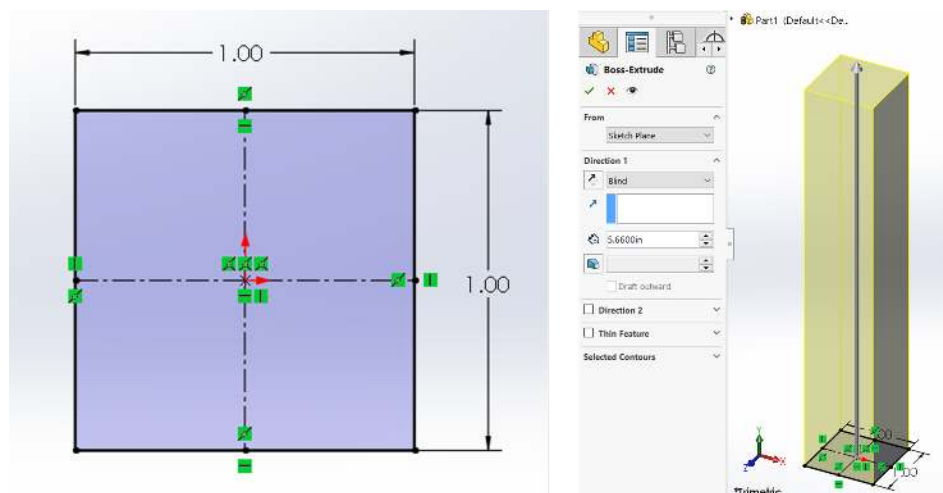


Figure 4.27 Base Extrude for CAE1002

After extruding the base rectangular shape, sketch on the right face of the rectangle a 5.25 by .625 rectangle. Use center lines from the midpoints of the rectangular face to center the rectangle using vertical and horizontal relations. Add .125 inch sketch fillets to the corner of the rectangle sketch and cut extrude to a depth of .050 inches.

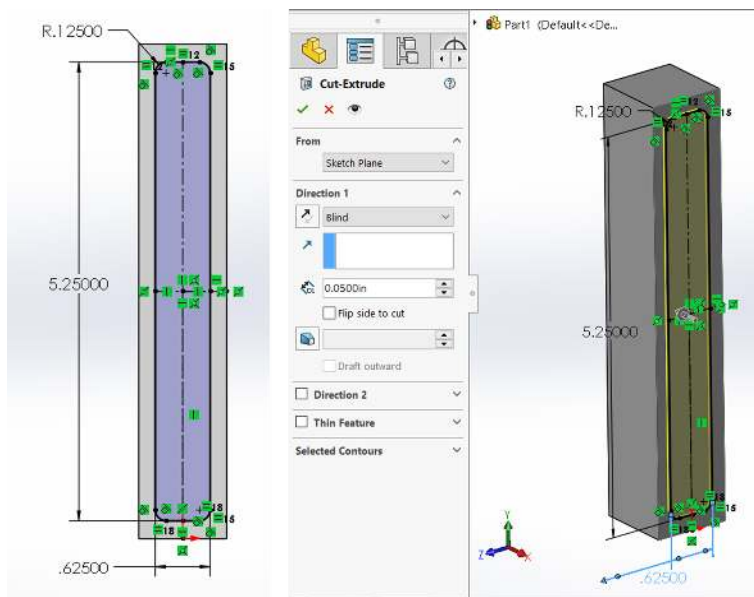


Figure 4.28 Cut Extrude on Right Side

Once the cut extrude is done, mirror it to the left side of the part using the Right Plane as the mirror plane. If using the front plane isn't working properly for you, you probably created the feature on the wrong face and should either edit the sketch plane of the original sketch or just use the Front Plane for your mirror plane.

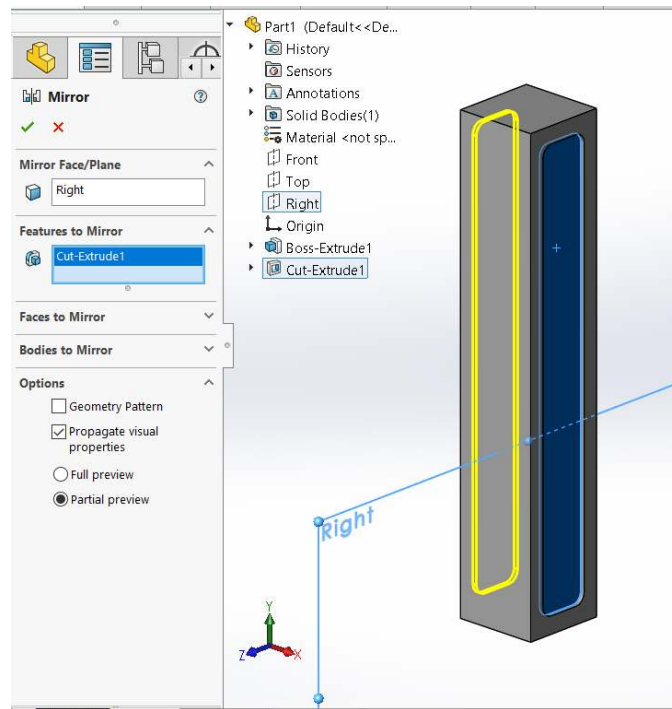


Figure 4.29 Mirror Pocket

Now start a sketch on the front face of the part, sketch the .3125 inch circle and the two .090 inch circles. Use a center line through the front face vertically to help you place and constrain the holes. It is recommended that you use a symmetric or mirror with the .090" holes.

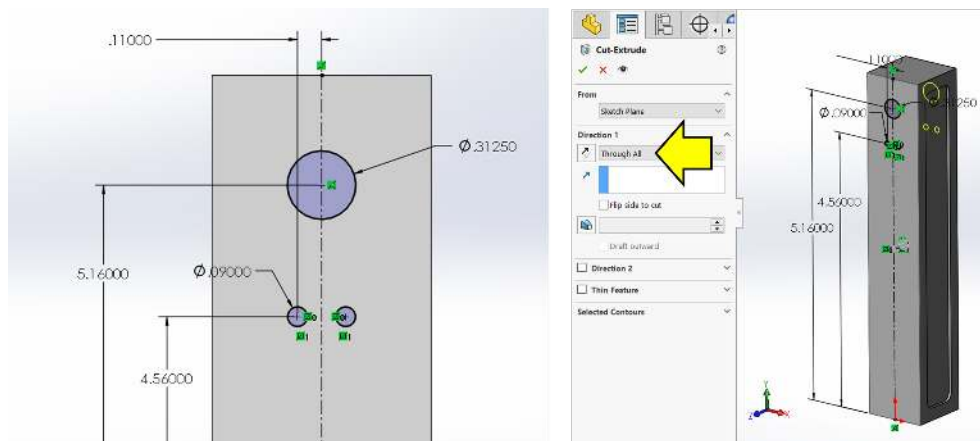


Figure 4.30 Front Hole Cut Extrude

HOLE WIZARD HOLES

For this part, there are many hole wizard features and therefore will add many separate hole wizard features because all the standard holes here vary in type and size.

To begin, activate the hole wizard tool and choose a .5000 (1/2) fractional size simple hole that has a through all end condition and a .564 far side countersink. There is also a tolerance on the drawing for this hole that you can add at the bottom of the property manager if you would like.

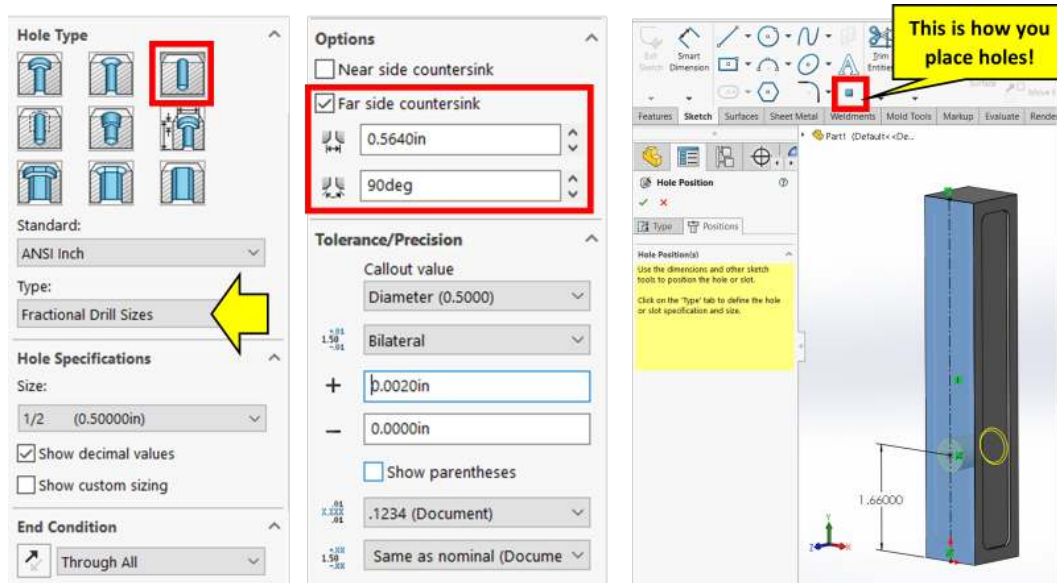


Figure 4.31 Half Inch Hole Type and Placement

Switch to the **Positions** Tab, select the front face of the rectangular part, but before placing the hole, activate the centerline sketch tool, draw a centerline through the middle of the front face, then activate the point command to place the hole on the center line. Dimension the hole from the bottom edge of the part to the center of the hole at 1.660 inches. Confirm the hole wizard.

Start another hole wizard feature for the 0.1875 (3/16) fractional drill size simple hole that has a blind end condition of .300 inches deep. To place the hole directly on top of the .090 inch hole, switch to the positions tab, select the front face, hover near the perimeter of the .090 hole to snap to its center.

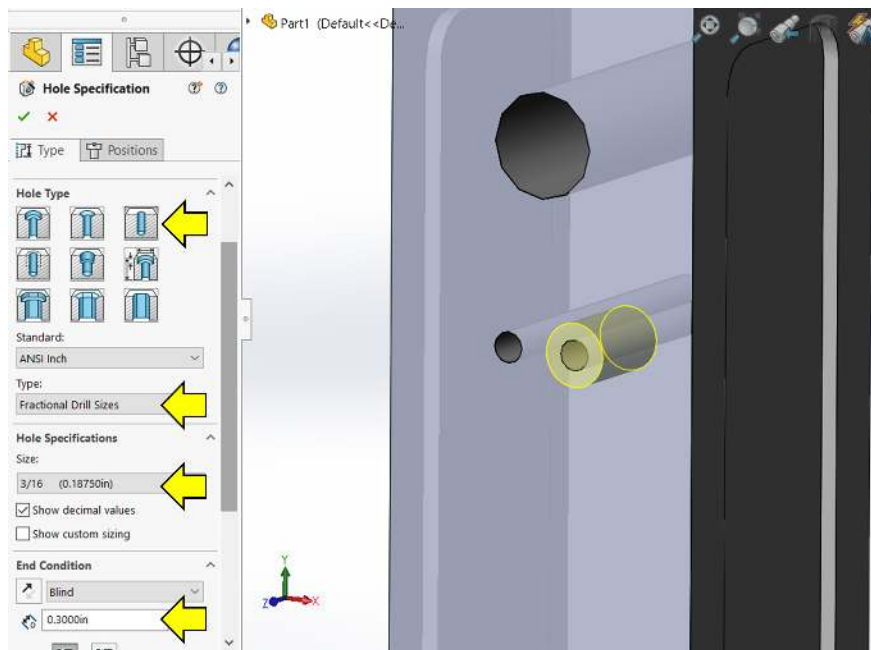


Figure 4.32 .1875 Hole Wizard Feature

Start another hole wizard feature for the bottom **1/4-20 tapped hole** that has a blind end condition of .500 for the threads and .7500 for the pilot hole. The tapped hole also has a near side countersink of .280 inches at 90 degrees. Place the hole using the positions tab on the bottom face directly in the middle of the face. The origin should appear when you select the face to help you place the hole in the middle.

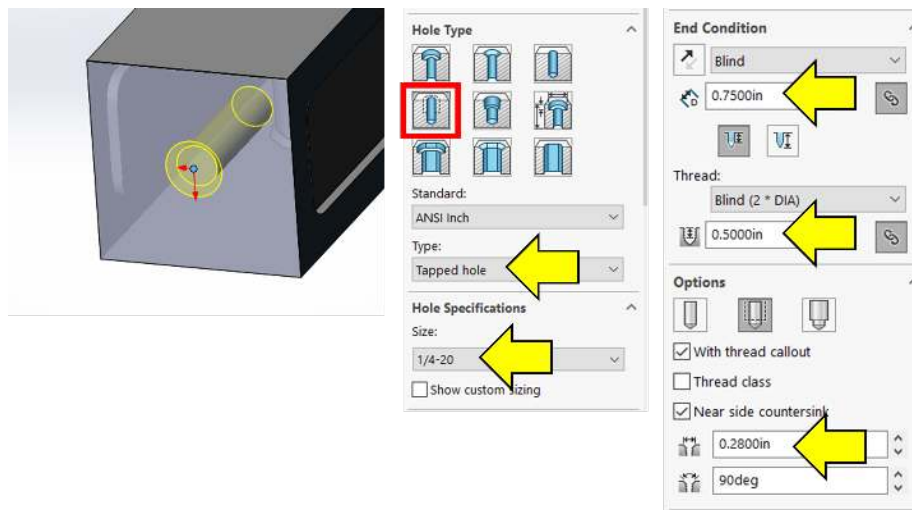


Figure 4.33 .25 Tapped Hole

Finally, using the same method as with the previous 1/4-20 tapped hole, place a **10-32 tapped hole** that goes up to the .3125 hole (use Up to Next) for both the pilot hole and threads. Also, make sure the tapped hole has a .220 inch nearside countersink at 90 degrees.

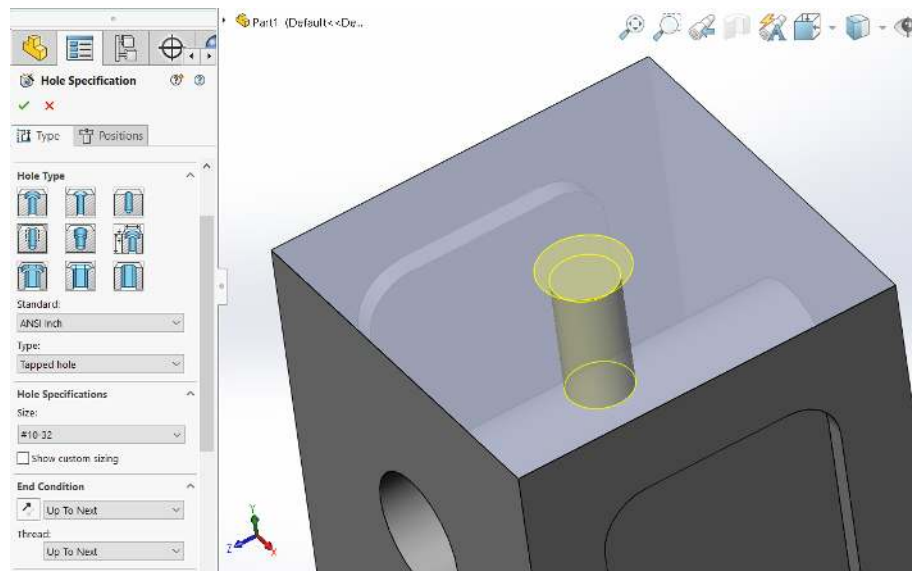
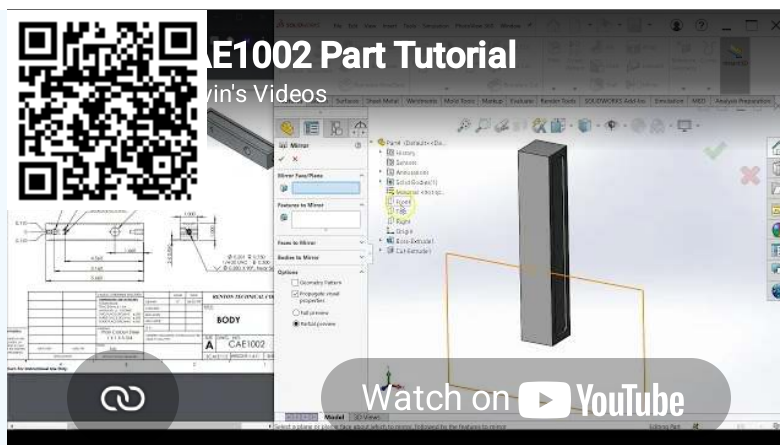


Figure 4.34 10_32 Tapped Hole

Now all that is left to do is specify the material as Plain Carbon Steel and save the part as CAE1002.

? CAE1002 Part Tutorial

If you struggled to follow along with the written instruction for this tutorial, follow along with the video tutorial below to complete your model. Also, begin to start practicing trying to figure out how to do a feature before it is instructed because eventually, you will be determining the method of modeling features in a mechanical design.



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CHAPTER OVERVIEW

5: Revolve and Other DFM Tools

Learning Objectives

The purposes of this chapter will be to...

- Use the **Revolve** features (Revolved Boss/Base and Cut) to model cylindrical parts for turning/boring and blow molding.
- Use the **Shell** feature and **Thin-Feature** Option to model uniform thickness parts for injection molding, casting, and sheet metal parts.
- Identify the functions of **Draft** and **Rib** features in efficiently aiding design of cast and injection molded parts.

In this chapter we will learn features that help us design parts for specific manufacturing processes such as turning/boring, casting, and plastic injection and blow molding. We never want to design part features based on manufacturing processes, just considering manufacturing processes available to us; this methodology in design is called Design for Manufacturing (DFM).

[5.1: Revolved Features](#)

[5.2: Shell](#)

[5.3: Draft](#)

[5.4: Thin Feature and Rib](#)

[5.5: CAE1004 Flywheel Tutorial](#)

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5.1: Revolved Features

Revolved features are used to primarily create cylindrical parts (though this doesn't mean they can't be used for other unique situations). Revolved features take the cross section of a cylindrical feature and then revolves it by a number of degrees (90, 180, 360, etc. See Figure 5.1).

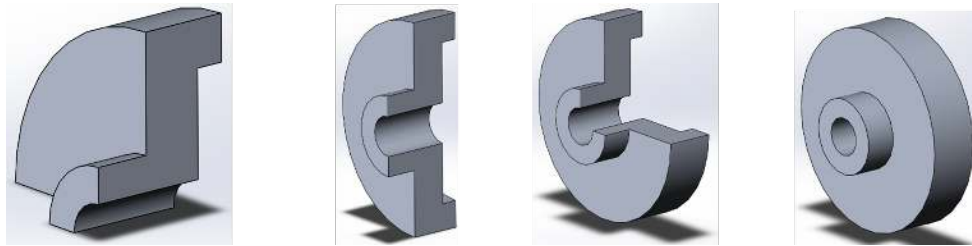


Figure 5.1 Revolved Sketch at Different Angle

The revolved part is generated from a sketch of the half section for the part (imagine a half section on a technical drawing). The sketch of revolved feature controls the cross-section of the feature and the revolved feature itself controls the axis of the revolution as well as how many degrees the profile is revolved.

To create a revolve feature, you first need to create the profile sketch. What the sketch needs to include depends on the scenario but one thing every revolve profile sketch needs is a centerline. The centerline is to tell the revolved feature where to revolve about (the axis of revolution). This should come as no surprise due to the fact that most all cylindrical parts have a central axis. It is good practice that the centerline for a revolved sketch be the very first thing you draw so that you can base the rest of that sketch off the centerline.

Revolved Feature sketches always need a centerline for specifying the axis of the revolved feature!

Because revolved features are used primarily for cylindrical parts that require lots of diameter dimensions to be specified, you can specify dimensions for diameters in the revolve profile sketch even though a circular feature doesn't exist yet. See the left image of **Figure 5.2** how the diameter dimensions reference horizontal lines and the centerline to dimensions implied diameters. The diameter symbol is automatically displayed and everything!

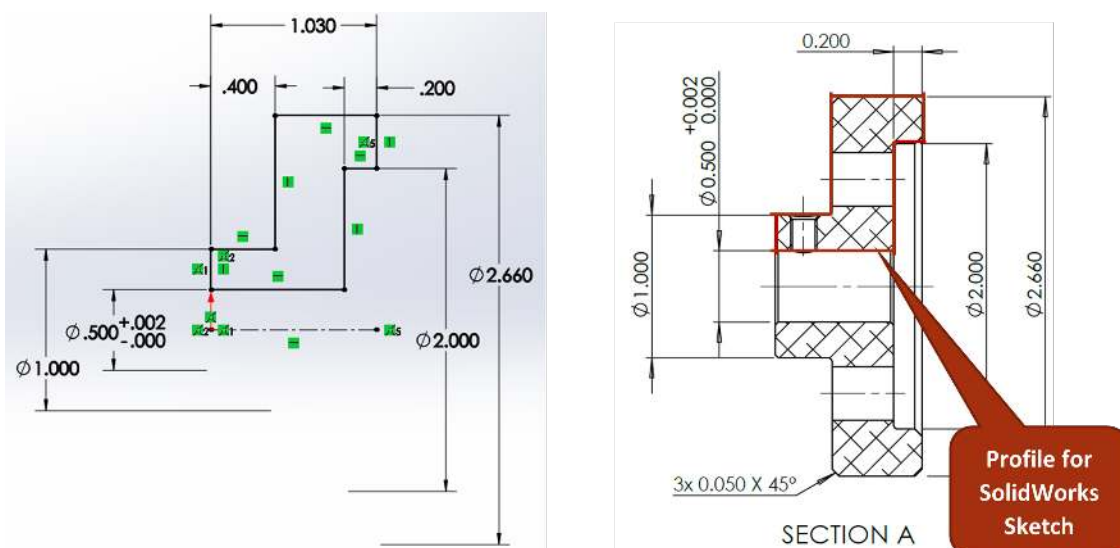


Figure 5.2 Creating a Revolved Sketch

To specify diameter dimensions for revolved sketch profiles, select the horizontal line that will create the diameter when the revolve is done as well as the centerline being used for the axis of revolution. The dimension will show as a radius until you move your cursor below the centerline. After specifying your first diameter dimension, selecting other horizontal lines defaults to a

diameter dimension. Eventually you finish specifying diameters and want to specify something else, so to end the automatic diameter dimension set, hit ESC from the keyboard or exit the command.

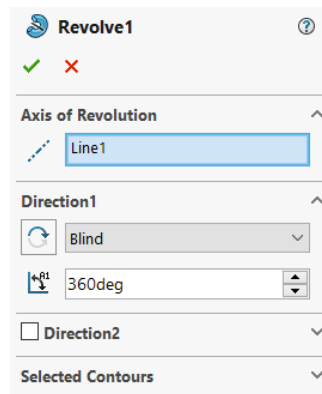


Figure 5.3 Revolve Property Manager

Once you have your sketch with the revolve profile and the centerline that will be used as the axis for the revolve, you can complete your revolve feature by activating Revolved Boss/Base from the Features tab of the CommandManager and using the property manager to specify your revolve settings (Figure 5.3).

? Exercise 5.1 Exploring the Revolve Feature

Here let's explore some of the components of a revolve sketch and the feature settings available for a revolved feature. Here we will first explore the Revolved Boss/Base tool and then also see how the Revolved Cut tool works along with the sketches involved.

1. Start a new sketch on the right plane. Sketch a centerline and a profile above the centerline.
2. Activate the Revolved Boss/Base Command, exploring the different options available.
3. Then, Create a new sketch also on the Right Plane of a simple circle on one of the edges of your revolved boss.
4. Finally, use revolved cut to cut the groove into the revolved boss using the existing axis.



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5.2: Shell

Shell is a great tool for parts requiring a uniform thickness because it is as easy as Fillet or Chamfer but can create some complex uniform thickness parts. Cast and Injection Molded parts are more producible and easier to manufacture if the thickness/cross-section of the parts is uniform and doesn't result in any drastic thickness changes.

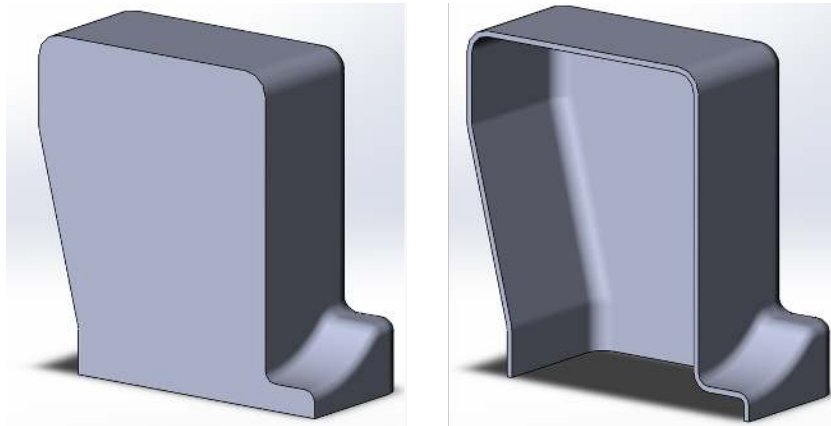


Figure 5.4 Cross Section of Before and After Shell Feature

To use the Shell command, all you need to specify is a face to be removed (in Figure 5.4, the bottom is removed) and a wall thickness. Shell is also a feature-based feature, so no sketch is required to use the shell command. However, you must have some sort of solid body in the model already before being able to use Shell.

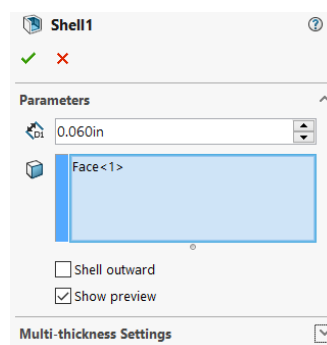


Figure 5.5 Shell Property Manager

When you activate the shell feature from the CommandManager, you select your face and specify your wall thickness. Note that you can select more than one face to create situations such as that shown in Figure 5.6.

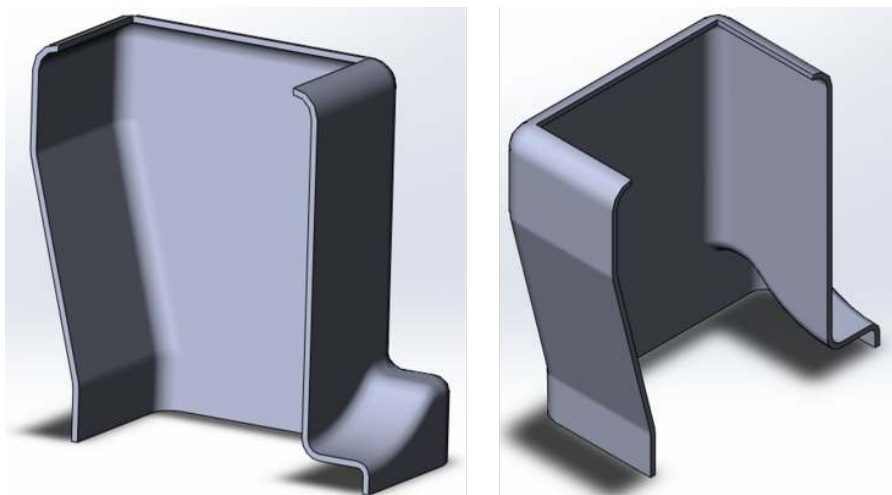


Figure 5.6 Removing Two Faces with the Shell Feature

One thing to be careful of with Shell is proper placement of the feature in the FeatureManager Design Tree. Placed too early or too late and the feature isn't appropriate for your scenario but placed just right will allow you to get uniform wall thickness. For the example used in Figure 5.4 and 5.6, the Shell was placed after all the Fillets to ensure that the Fillet radii would be offset in the Shell. However, it was made sure that Cut-Extrudes be made after the Shell to ensure proper geometry was obtained (Figure 4.7).

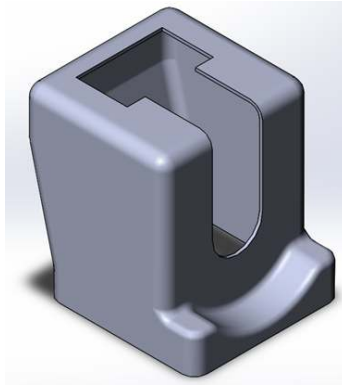


Figure 5.7 Shelled Part Showing Importance of Order of Features

? Exercise 5.2 Using Shell in Design

For this exercise, we will start with a model that has been prepped for using shell and perform the shell command.

1. Download the [CAE1009 Starter Part](#).
2. Roll the part back before the Shell using the rollback bar,
3. Shell the part with a uniform wall thickness of .060 inches and removing the bottom face. Also feel free to practice placing cut extrudes after the shell using the “Up to Next” option.



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5.3: Draft

Draft is used to make straight, vertical surfaces angled. This is useful mainly for cast parts where “Draft Angle” is key to production capability of a part. For green sand cast parts to be manufactured to high quality, draft angle needs to be placed on all vertical sides so that the pattern for the part and mold can be set without any cracks or crumbles in the fragile sand mold.

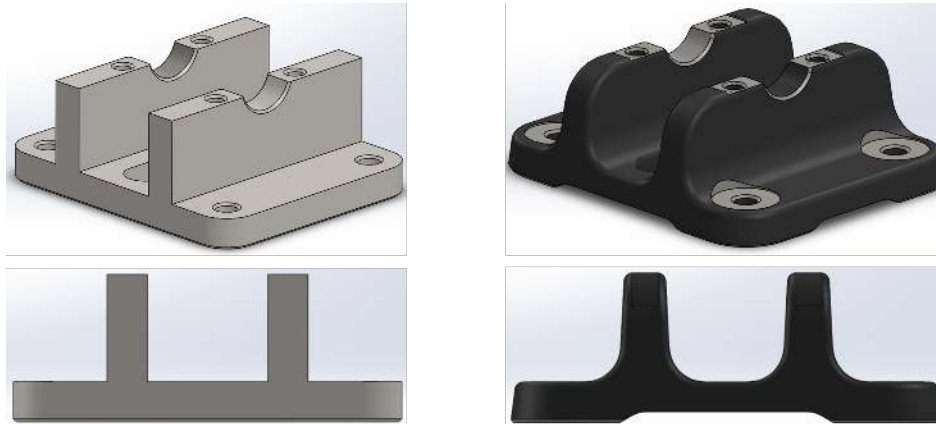


Figure 5.8 Part Concept Without Draft for Machining [left] and With Draft for Casting [right]

Draft angle is used by selecting faces and specifying a draft angle after activating the command from the CommandManager. There are more advanced options that allow you to choose where the parting line is as well. An example of Draft Angle being used on a machined to cast conversion is shown in Figure 5.8.

? Exercise 5.3 Using Draft Features

Download the [Machined/Cast Conversion part](#) and apply a 7 degree draft to all outer vertical faces and a 3 degree draft to the shaft support vertical faces as indicated. Follow along with the video before for more detail on what to do.



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5.4: Thin Feature and Rib

THIN FEATURE

Thin features are an option for most sketch-based features such as Boss Extrudes, revolves, sweeps, etc. Rib is automatically a thin feature because it creates a 3D solid from an open sketch. Thin-features are features that extrude in 2 directions: the first direction is perpendicular to the sketch, which is normal for any 3D sketch-based feature, and the second direction is offset from the original sketch giving the sketch a thickness to be extruded.

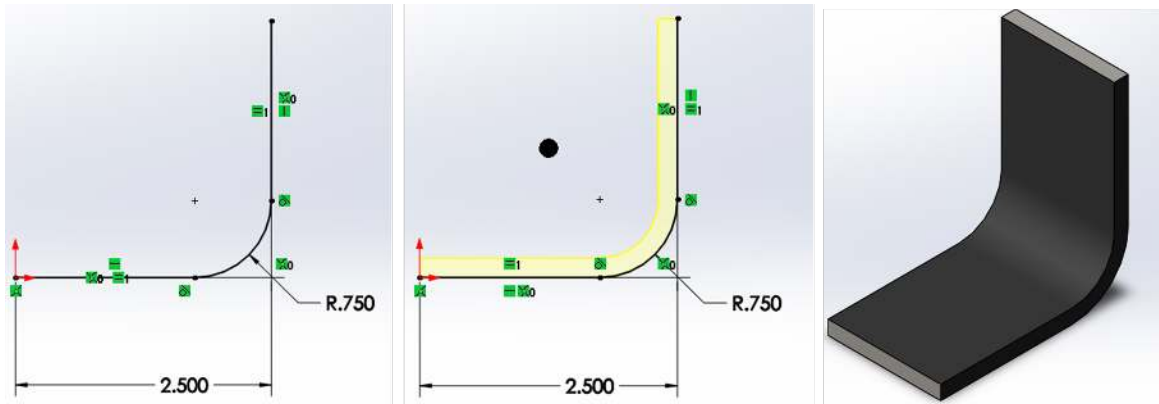


Figure 5.9 Process of Creating a Thin Feature

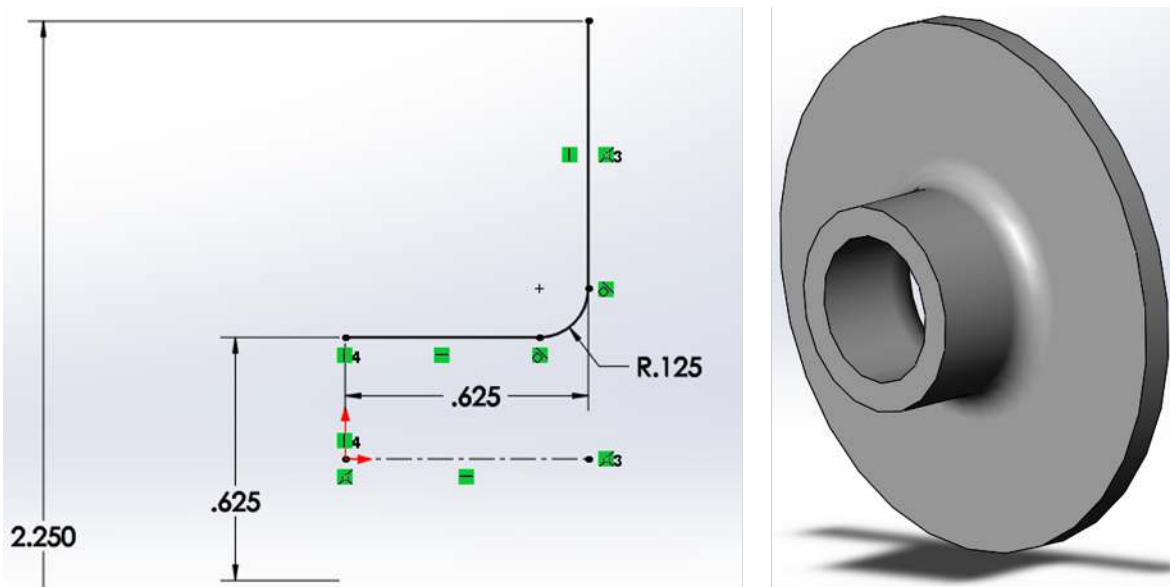


Figure 5.10 Revolved Thin Feature

To create a thin feature, you create a new sketch, sketch the open sketch profile of your object, activate the feature you wish to create, and be sure to check the “Thin-Feature” Option near the bottom of the property manager for the feature.

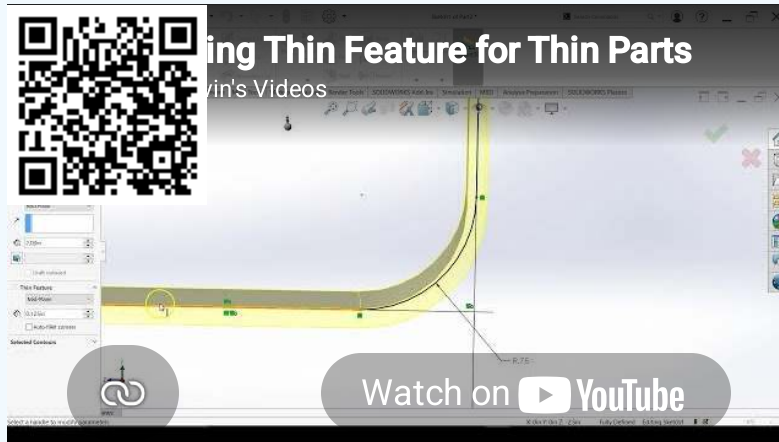
If a dialog box comes up asking if you want to close the sketch, say NO!

? Exercise 5.4 Creating a Thin Feature

Here you will explore creating thin features from Extruded Boss Base as well as Revolved Boss Base.

- Start a new part file in SolidWorks,
- Sketch on the Right Plane a fully-defined profile such as is shown in figures 5.9 and 5.10.

- Activate Extruded Boss/Base and choose Thin Feature. Specify the thin feature parameters, along with a MidPlane end condition and confirm.
- Move the Rollback bar before the Extruded Boss/Base and now try with Revolved Boss/Base and the same sketch. Cancel when you have finished experimenting with the different options.



RIB

Rib is a sketch-based feature like the Extrude and Revolve tools but extrudes in 2 directions at once to create a support feature for unstable shapes. Ribs are very common in cast and injection molded parts because adding them better allows for maintaining a uniform wall thickness throughout the entire part, which is good for casting and injection molding manufacturing processes.

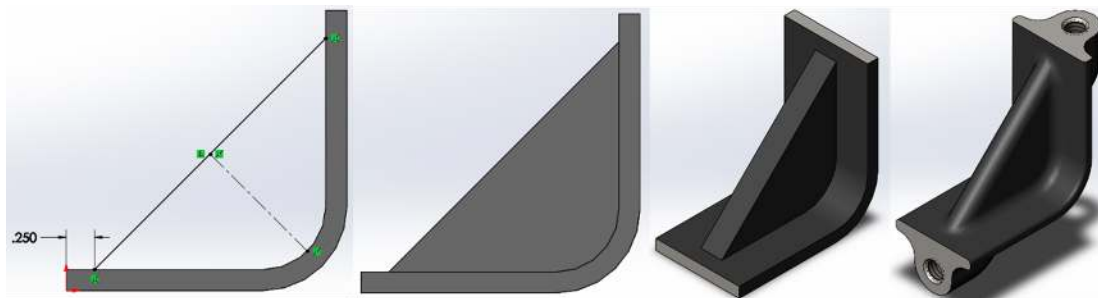


Figure 5.11 Rib Process Example

To create a Rib feature, you start a sketch on a plane that is located at a point where you want to place the rib, draw a open sketch of the shape you want to make a rib from and then activating the rib feature creates the rib with the specification of thickness, extrusion direction, and other advanced options.

? Exercise 5.5 Creating Rib Features

Using your same sketch from the last exercise, create a thin feature using Extruded Boss Base and then add a rib feature to reinforce the bracket. See the finished product as well: [2130004_RibEx](#)

- Create the bracket part from Exercise 5.4 with a Extruded Boss Thin Feature using the midplane option for extrusion.
- Create a new sketch connecting the faces of the bracket on the Right Plane.
- Activate the Rib tool straight from within the sketch and specify the rib parameters as shown in figure 5.11.



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5.5: CAE1004 Flywheel Tutorial

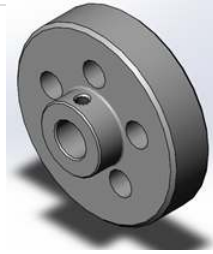


Figure 5.12 Finished CAE1004 Flywheel

Open a new part in SolidWorks and start a new sketch on the Right Plane by right clicking on the right plane and selecting the sketch icon. [CAE1004_Flywheel](#) was designed in inches so be sure your units are set to inches.

THE REVOLVE

Draw a horizontal centerline from the origin to the right. The length of this line does not matter.



Figure 5.13 Centerline for Revolve Sketch

Sketch above the centerline a rough outline of the top portion of the section view shown in the drawing excluding all holes. Make your sketch as proportional as possible to the actual geometry seen in the drawing. The sketch should look like a blocky letter “S”.

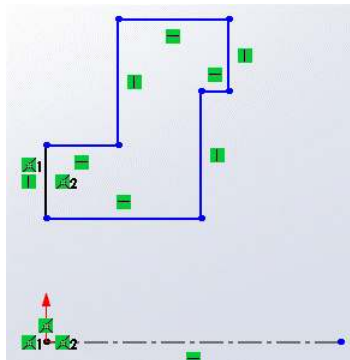


Figure 5.14 Rough Sketch for Revolve

Place diameter dimensions by selecting each horizontal line of the sketch and the center line. When your cursor is above the centerline, the dimension will only show as a radius. But when you move your cursor below the centerline, the dimension becomes a diameter. Here we will add the .5-inch diameter, the 1-inch diameter, the 2-inch diameter, and the 2.66-inch diameter.

In this sketch still, also add the .2-inch width, .400-inch width and the 1.030-inch width by selecting the vertical lines that represent the width.

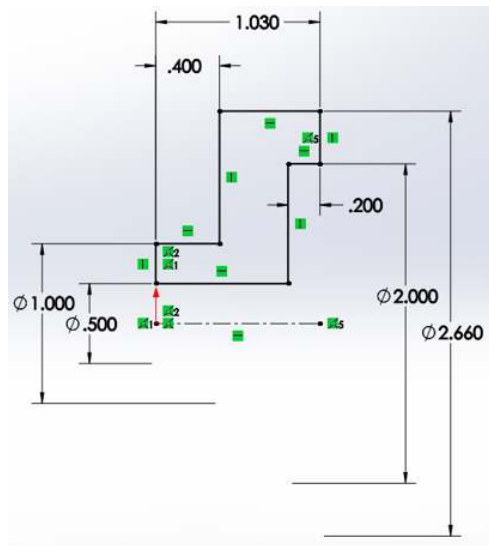


Figure 5.15 Fully Defined Revolve Sketch

Ensure that your sketch is fully defined after adding these dimensions (except for the center line endpoint which will remain underdefined. This is acceptable but if it really bothers you like it does me, you can add a relation between the centerline endpoint and the overall width edge of the sketch.

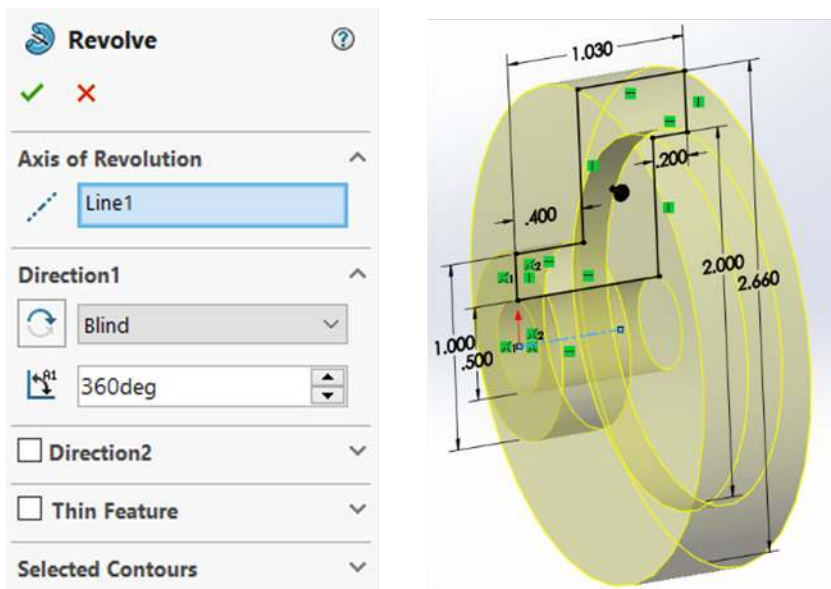


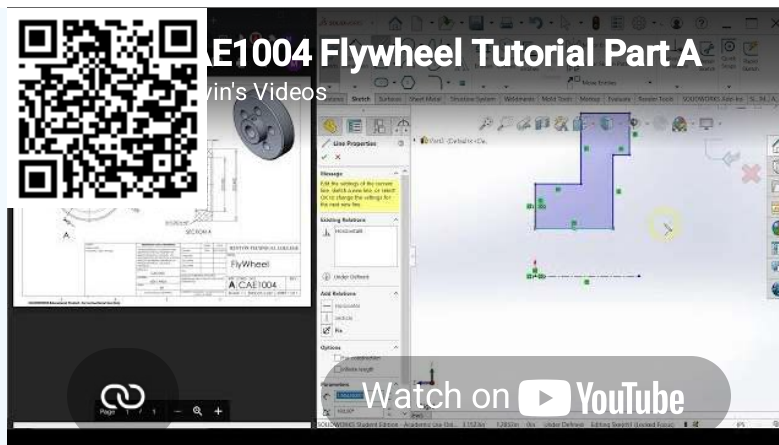
Figure 5.16 Revolve of Flywheel Sketch

Select the Revolved Boss/Base tool from the features tab and then confirm the feature. You should see a yellow preview of the revolved feature revolving 360 degrees the profile of the sketch we just drew.

For revolve, a centerline axis and sketch is required. If there is only 1 centerline in your sketch, revolve will automatically select that one. Otherwise, you specify the centerline manually into the axis selection box of the Revolved Boss/Base property manager.

? Exercise 5.6 The Revolve of CAE1004 Flywheel

Start a new part and complete the revolve feature of the CAE1004 Flywheel.



HOLE WIZARD HOLES

Start the hole wizard feature command from the CommandManager and specify a **10-32 Straight Tapped Hole** that goes up to the next surface (Up to Next) and a nearside countersink of .220 inches by 90 degrees (Figure 5.17).

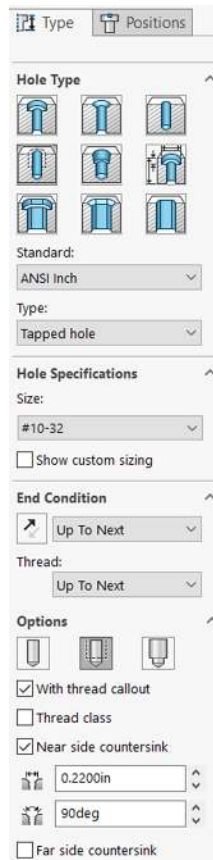


Figure 5.17 10-32 Tapped Hole Type Settings

Switch to the positions tab and select the 1-inch outer diameter face as the face to place the hole on. Place the hole near where we desire it, draw a centerline from the front edge to the back edge of the 1-inch diameter face. Click the centerline and in the context toolbar, specify an “**Along Z**” relation. Then CNTRL-click the front endpoint of the centerline and the origin and specify an “Along Y” relation. Finally, add a midpoint relation to the centerline and the point for the tapped hole. Confirm the hole wizard (Figure 5.18).

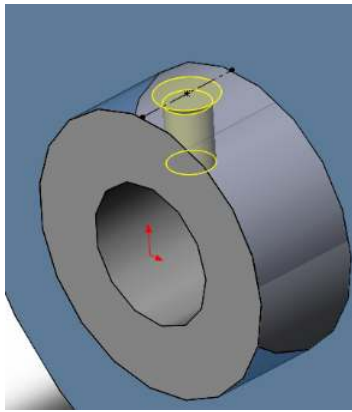


Figure 5.18 Positing 10-32 Tapped Hole

Start the hole wizard a second time and specify a **3/8 of an inch hole (.375)** that goes through the entire part. Switching to the Fractional Drill Size hole type helps find the hole size faster. Switching the “Show Decimal Values” also helps when you aren’t sure of the drill size.

Switch to the positions tab, click the back inside face of the part, activate the circle sketch command and place a 1.5-inch construction circle at the origin of the sketch. Remember that you can change the circle to construction from the property manager (“For Construction” Option) as well as by clicking the circle and selecting “Construction Geometry” from the context toolbar.

Activate the point command again to place the .375-inch hole at the top quadrant of the construction circle and confirm the hole wizard command.

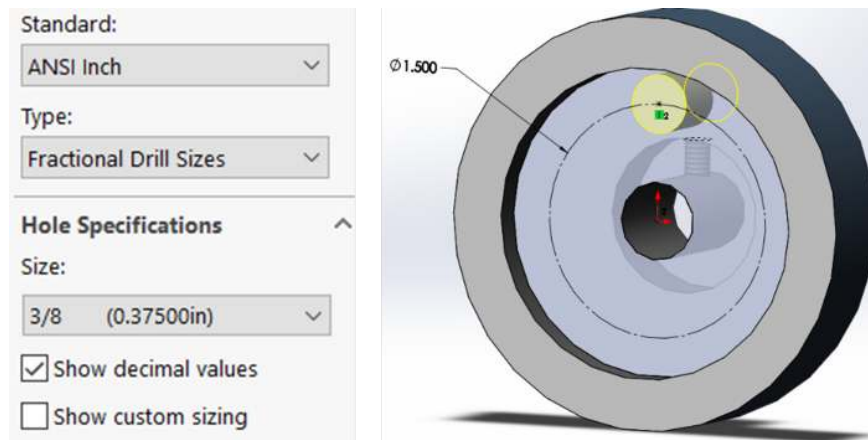


Figure 5.19 Specifying the .375 Hole

Activate the **Circular Pattern** command from the features tab of the CommandManager and pattern the circle 5 times equally spaced around the center of the part. To do this, be sure that the 3/8 of an inch hole is selected in the “Features and Faces to Pattern” selection box and the center hole of the part is selected in the “Pattern Axis” selection box of the property manager. Confirm the circular pattern command.

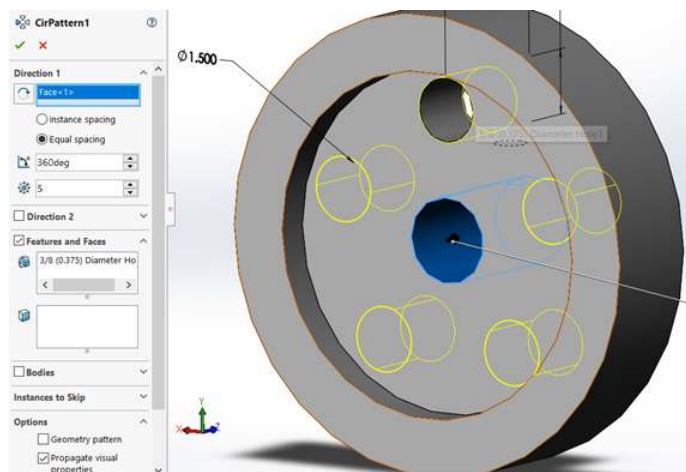


Figure 5.20 Circular Pattern of .375 Hole

? Exercise 5.7 Adding the Holes to CAE1004

Add the 10-32 tapped hole and .375 through hole using the Hole Wizard and Circular Pattern features.



FILLETS AND CHAMFERS

Add a **.032-inch fillet** using the fillet feature from the CommandManager to the two internal corner edges of the part as shown in the drawing.

Add a **.050-inch by 45-degree chamfer** using the chamfer feature from the fillet drop down in the CommandManager to the edges of the 2.66-inch diameter face and the back edge of the 2-inch diameter pocket. If you don't see the chamfer preview, check to make sure the "Full Preview" option is set in the property manager.

To finish our part, add a **.020-inch by 45-degree chamfer** to the internal edges of the .5-inch diameter hole and the front edge of the 1-inch diameter face.

Set the material as 6061 Alloy and save your part as CAE1004.

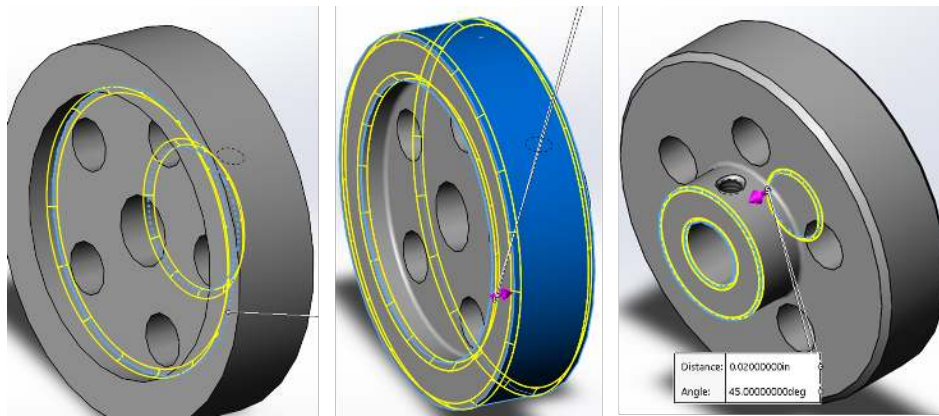


Figure 5.21 Fillets and Chamfer Selection for Flywheel

? Exercise 5.8 Finishing Up CAE1004 with Fillets and Chamfers

Add the remaining fillets and chamfers to the specified edges of CAE1004



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CHAPTER OVERVIEW

6: Complex Shapes

Learning Objectives

By the end of this chapter, you will be able to...

- Create reference geometry to be used in feature creation.
- Use sweep, helix, loft, and wrap on a basic level to create complex features.

This chapter is all about exploring the SolidWorks tools that will allow you to create complex shapes. The world isn't filled with just boxes and cylinders, its filled with some weird stuff that is specifically shaped to accomplish an increased functionality.

- Imagine flexible piping or guard rails; how would you model those in SolidWorks?
- Imagine bicycle frames and vacuum nozzles; how would you model those in SolidWorks?
- What about everyday mechanical components like springs, threads, and cams?

This chapter will give you the introductory knowledge and skills to model all the things listed above using 4 key features in SolidWorks:

- Reference Geometry
- Sweep and Helix
- Loft
- Wrap

[6.1: Reference Geometry - Planes](#)

[6.2: 2130005A Simple Camshaft Tutorial](#)

[6.3: 2130006 Bike Frame Tutorial](#)

[6.4: Sweep](#)

[6.5: Loft and Wrap](#)

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6.1: Reference Geometry - Planes

Reference geometry is geometry that we create to be used as reference for the creation of new sketches and features. The types of reference geometry that exist are planes, axes, coordinate systems, and points. We will save axes, coordinate systems, and points for you to learn on a case by case basis when you get more familiar with SolidWorks. Reference planes are certainly the most commonly used reference geometry (besides reference sketches). There are many types of reference planes but here we will focus on the commonly used.

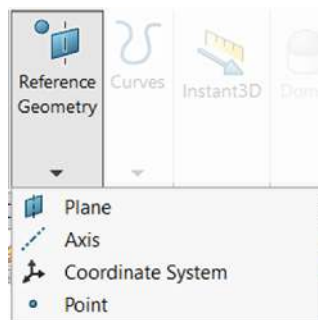


Figure 6.1 Reference Geometry Icons

The reference geometry tool is located to the right of the Features Tab of the CommandManager. When you choose the dropdown of the command, you will see icons for each of the types of reference geometry that you can select.

PLANES

As was noted earlier, reference planes are the most common reference geometry used in part creation. You need reference planes for extruding anything at an angle, for creating lofts, and usually for creating sweeps. Reference planes also help in speeding up the modeling process.

For example, sometimes you need to create a reference plane to be the mirror plane for a mirror feature; mirroring the features instead of recreating them is of course much faster. When you have offset extruded or revolved features, it is very quick to create a reference offset plane to be your sketch plane for extruding the offset feature.

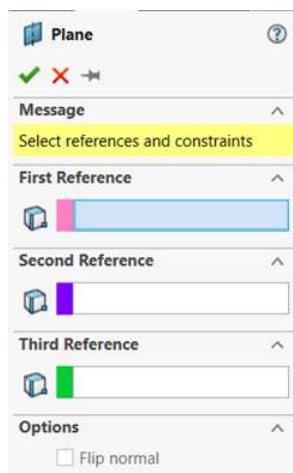


Figure 6.2 Reference Planes Property Manager

How the reference planes work is as soon as you activate the reference plane command, the property manager appears with three selection boxes for a first reference, second reference, and third reference. You can have up to three references to create a reference plane, but some reference planes may only require one or two references. As you select references, options become available for how to constrain the reference plane based on your references. Below, we will walk through the most practical ways to use reference planes.

Common options for reference planes can be seen theoretically from the [SolidWorks Help on Planes](https://solidworks.com/learn/3d-modeling/reference-planes).

OFFSET PLANE

The offset plane option is selected by default if you select a plane or face for a first reference. The most common use of offset planes is for creating features that are offset from one of the default planes, for loft, wrap, or swept features.



Figure 6.3 Various Camshaft Designs (Source: [Wikipedia](https://en.wikipedia.org/wiki/Camshaft))

A great example of using an offset plane is in the modeling of a camshaft. When you first revolve the bearing surfaces and main body of the camshaft, you then need to locate a point where the first lobe will go. This is done with an offset plane, where a profile sketch can then go.

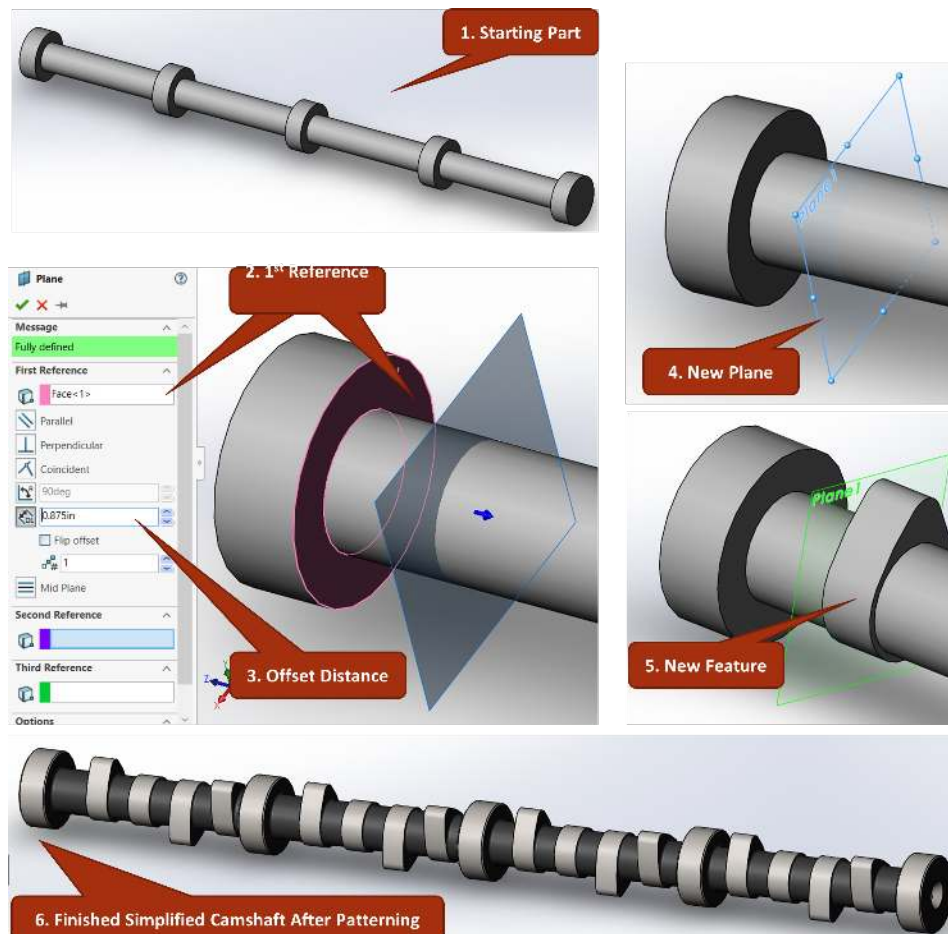


Figure 6.4 Using Offset Plane for Creating a Camshaft Lobe

MIDPLANE

The midplane option for the reference plane is most commonly used to create a mirror plane for when you want to mirror something that isn't symmetrical across the default planes. To specify a midplane, you choose two opposing surfaces as the first and second reference. As shown in Figure 6.5, they don't even have to be parallel or planar. As soon as you select the second reference, the reference plane property manager will automatically switch to midplane. It will show as an offset plane until the second reference surface is selected.

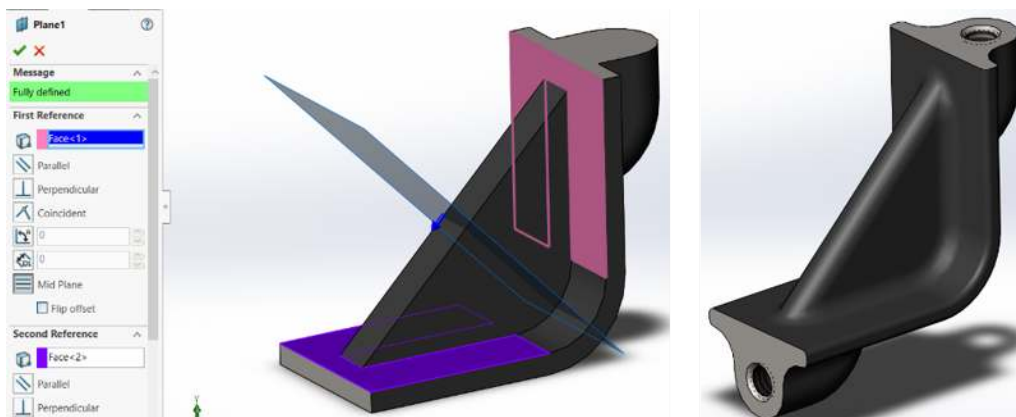


Figure 6.5 Using Midplane on Non-Parallel Surfaces to Establish Mirror Plane

Let's look at the camshaft again to see how the midplane option of reference plane was used in this part. The origin is set at the left most point of the part due to **geometric dimensioning and tolerancing** schemes so the middle of the shaft's main body is not lined up with the default planes. Therefore, to mirror the countersunk hole on the right side to the left side, it is required to create a midplane feature at the center of the shaft main body.

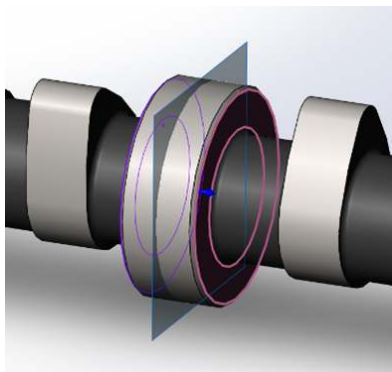


Figure 6.6 Selecting Two Parallel Surfaces for a Reference Midplane

? Exercise 6.1 Creating Offset and MidPlanes

In this exercise, we will practice creating offset planes and midplanes as well as exploring the different options of each.

1. Create an 2 inch offset plane from the Front Plane.
2. Try the patterning feature and flip offset as well to see how you can make multiple offset planes that are an equal distance apart.
3. Create a midplane between two of the offset planes that you have just created.
4. Create a midplane between the Front and Top Plane and flip the offset to see the impact to the reference plane.



PLANES USING REFERENCE SKETCHES

To create models where there are different shapes coming from all different directions, such as piping or a bicycle frame, creating planes oriented to reference sketches brings the seemingly complex task down to simplicity. To create a plane that uses a reference sketch, the sketch must of course already exist. So, the first step to creating this type of plane is creating the reference sketch first, which is usually one or more centerlines that define the direction of the extrusion you desire. After having the reference sketch, you select different parts of the sketch to base your reference plane from, such as being perpendicular to a line and going through a vertex.

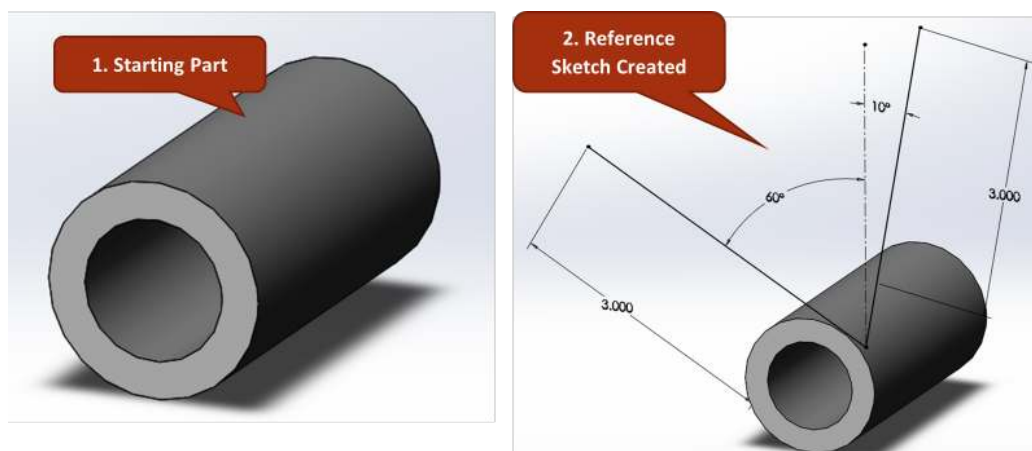


Figure 6.7 Creating Reference Sketch for Reference Planes

As noted previously, a bicycle frame is an excellent example where the directions of the tubing are at different angles. The base part is first created, then a reference sketch on a default plane that contains reference lines that will define the direction and length of the tubing members. After the reference sketch is made; the reference planes are created to pass through each end point of the reference line of the reference sketch as well as be perpendicular to the reference line. With the two new reference planes created, boss-extrusions can be made in the profile shape of the tubing (two concentric circles) extruded up to the next surfaces (using the Up to Next) option.

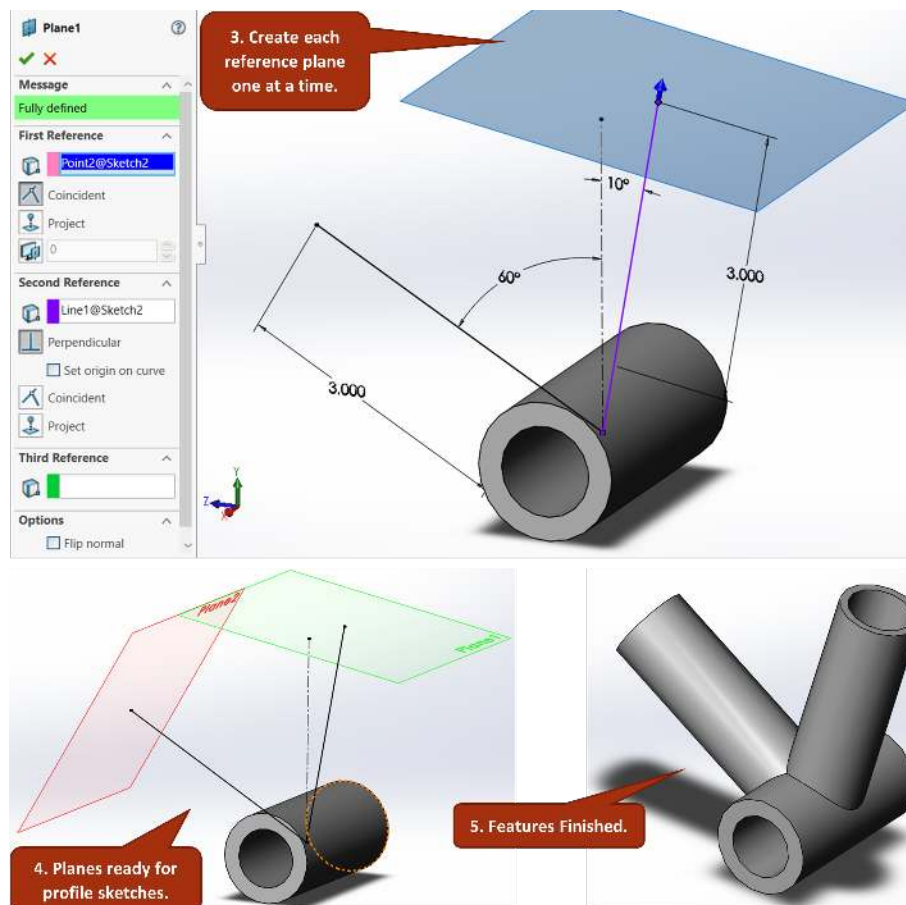
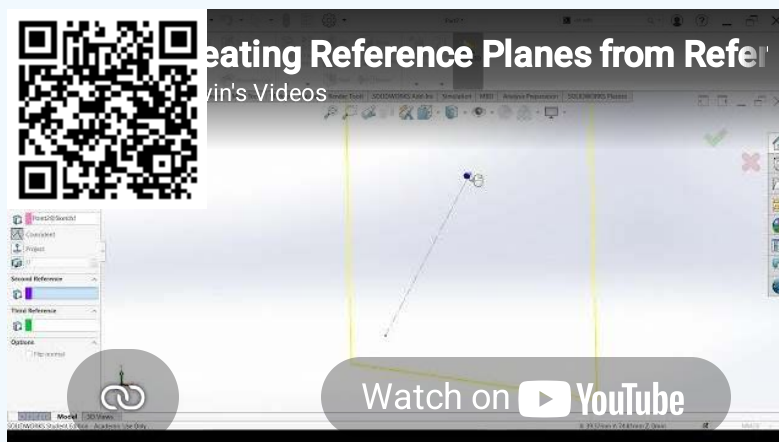


Figure 6.8 Creating Reference Planes from Reference Sketch for Bicycle Frame

? Exercise 6.2 Creating Planes from Reference Sketches

In this exercise, we will create a simple reference sketch such as the one shown above and then create reference planes from them.

- Create a sketch of at least one angled line on the front plane and exit the sketch.
- Activate the Reference Plane tool and create a reference plane by selecting the end point of the line and the line itself and confirm the reference plane.



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6.2: 2130005A Simple Camshaft Tutorial

In this tutorial, we will model **2130005A** which will require the use of the midplane and offset reference geometry tools. You will also use a new sketch tool called **Convert Entities** in this tutorial. To help you follow along with this written tutorial, there are also video tutorials throughout this section.

BASE REVOLVE & OFFSET PLANES

To start this part, open SolidWorks and start a new part with IPS units and ANSI Standard set. **Start a new sketch on the Right Plane** in preparation for a revolved feature.

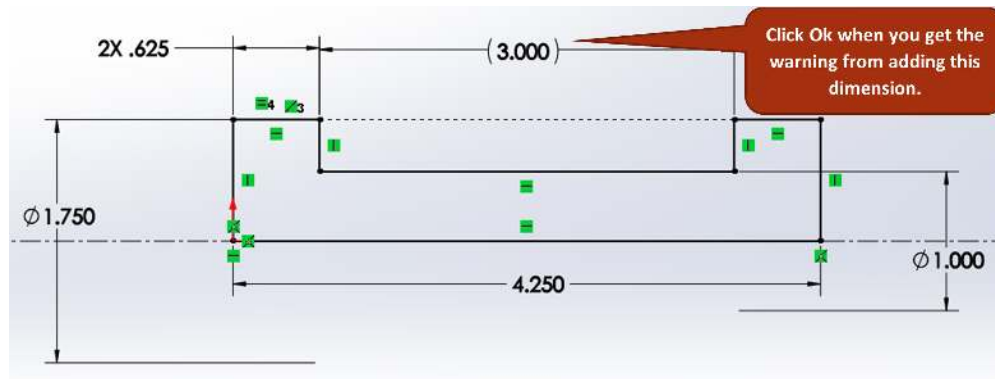


Figure 6.9 Sketch for Base Revolve of 2130005A

Sketch the top half of the outline of the part shown in the right view of the drawing without the cam lobes (see figure 6.9). Remember that you need a centerline in your sketch. The one for this part goes best horizontally from the origin. You can also try using the infinite line option.

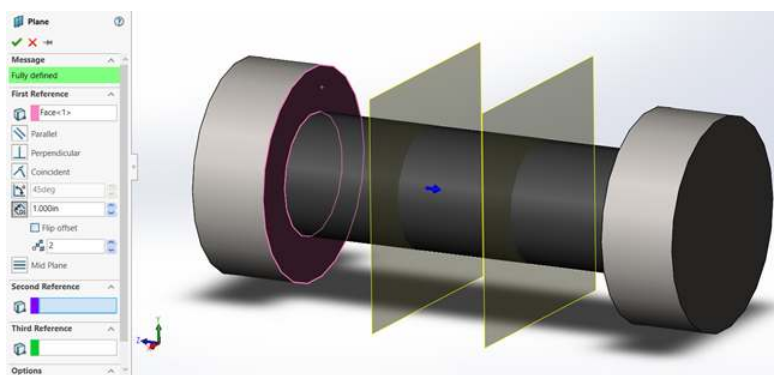
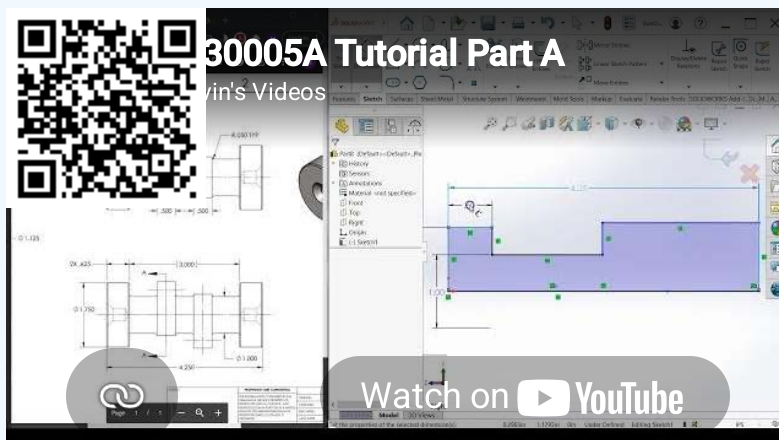


Figure 6.10 Creating Offset Planes for 2130005A

? Exercise 6.2 Base Revolve and Offset Planes



LOBE SKETCHES AND EXTRUDES

Start a new sketch on the leftmost plane of the two you've just created and draw the sketch shown in figure 6.11. You should start with a 1.125" diameter circle at the origin, a smaller 7/32" radius circle directly above it (use a vertical centerline to ensure the two circles are vertical). Then draw an arc on one side that is tangent to the 2 circles and has a radius of 2-inches.

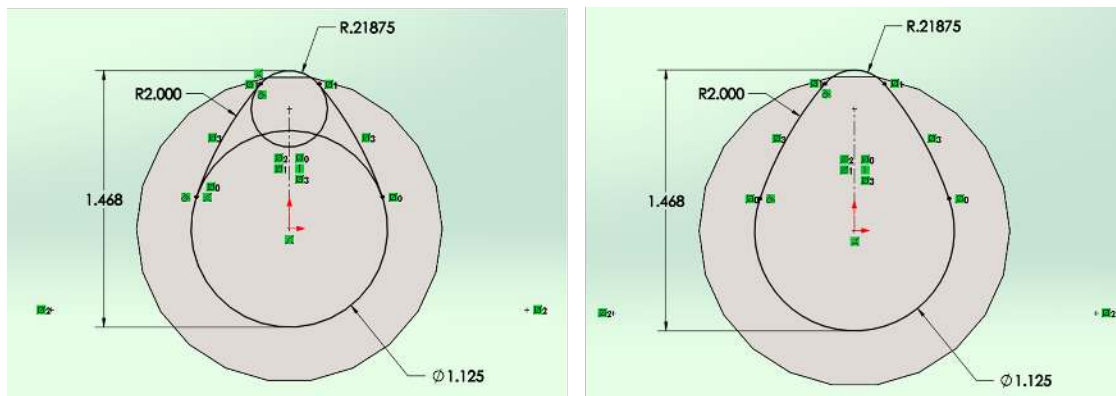


Figure 6.11 Cam Lobe Sketch Sequence for 2130005A

You can then **mirror the arc** to the other side using the vertical centerline. To get the 1.468 dimension to go to the maximum arc distances, you will have to **select the dimension**, switch to the leaders' tab of the property manager, **scroll down** to the bottom and switch both the first and second arc conditions to max.

When everything is fully defined, **trim away** the excess. Trim in a chain sequence as to not eliminate any relations that you added. **Extrude** the cam lobe using the midplane option for the boss extrude direction to .500". **Confirm** the boss extrude.

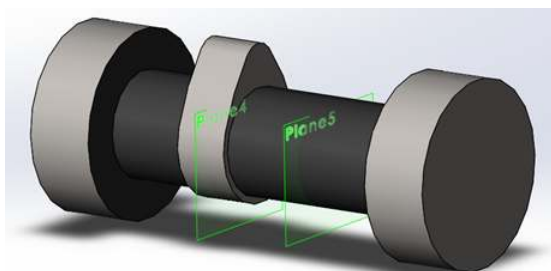


Figure 6.12 Extrusion of Cam Lobe

Start a new sketch on the right reference plane of the two you added, click the right surface of the cam lobe you've just created, and click the "Convert Entities" button from the Sketch tab of the CommandManager. The outline of the cam lobe should now appear on your new sketch plane. **Convert** all the entities you've just retrieved with Convert Entities to Construction Geometry (figure 6.13).

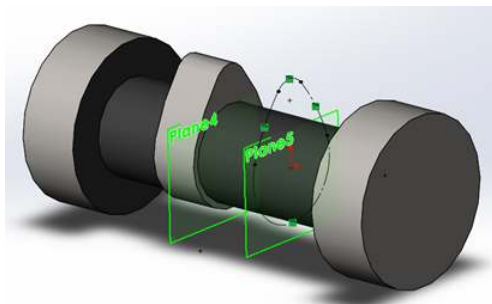


Figure 6.13 Convert Entities of Lobe for Second Lobe Sketch

Draw a horizontal centerline from the origin of the sketch to the right, **activate the mirror** entities command and mirror the lobe sketch entities across the horizontal centerline. After performing the mirror, **convert** the mirrored entities to solid lines, then **extrude** to 0.500 inches using the extrude midplane option (Figure 6.14).

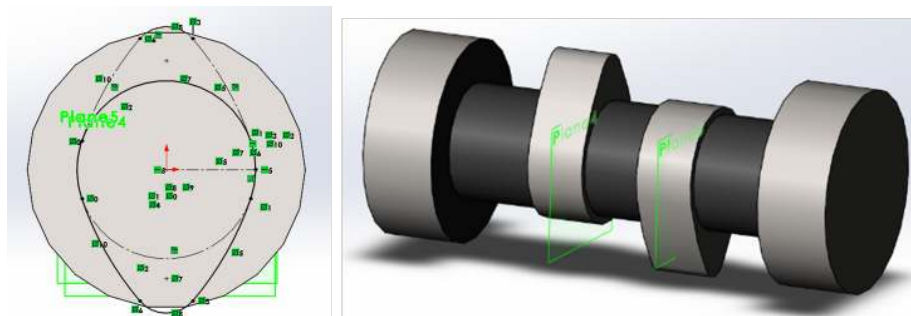
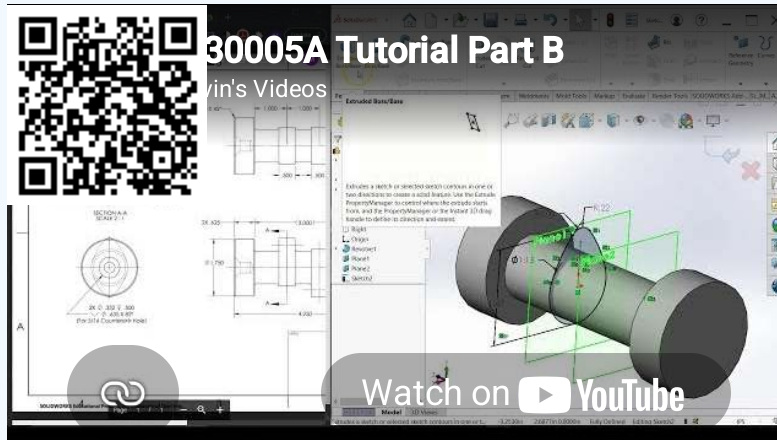


Figure 6.14 Mirror Lobe Sketch and Extrude

? Exercise 6.3 Lobe Sketches and Extrudes



REFERENCE MIDPLANE, HOLES, FILLETS, CHAMFERS

Create a **reference midplane** using the right most and left most faces of the camshafts. This will put a new plane directly in the middle of the part (figure 6.15).

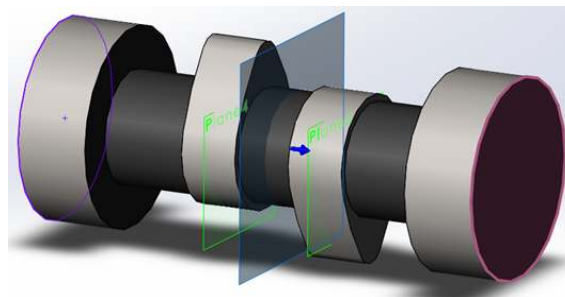


Figure 6.15 Add Midplane to the Camshaft Using a Reference Plane Feature

Then **activate the hole wizard** to place a 5/16" countersunk hole on the right end face of the camshaft directly in the middle on the origin having a depth of .500-inches. Use the **mirror feature** and the new midplane created to mirror the countersunk hole wizard hole to the other side.

Hide all reference frames by left clicking them and choosing the eyeball with a slash through it from the context toolbar that appears as you select each reference plane.

Add the fillets and chamfers using the 3D fillet and chamfer tools as specified in the drawing, the material, and save.

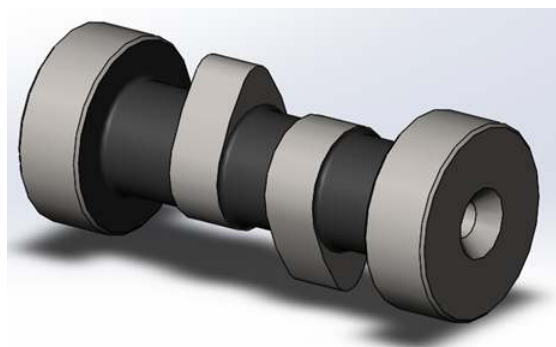
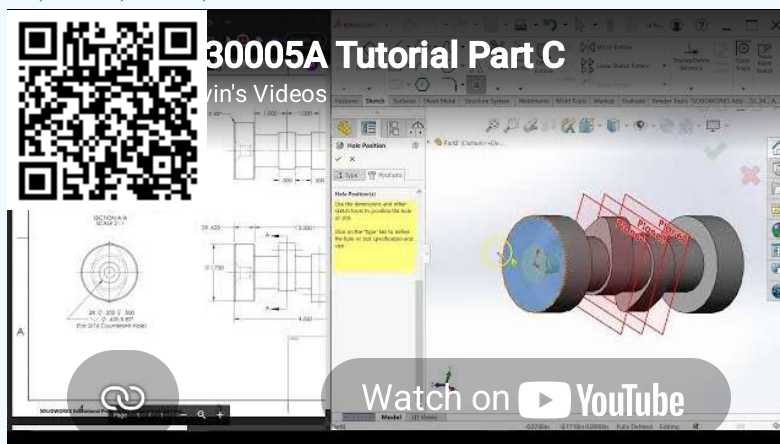


Figure 6.16 Finished 2130005A

? Exercise 6.4 Midplane, Holes, Fillets, and Chamfers



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6.3: 2130006 Bike Frame Tutorial

In this tutorial we will model [2130006 BikeFrame](#) which will require the creation of reference planes based on a reference sketch. The sketches will be simple – two circles a couple lines at most – but because of the capabilities of reference planes, we will create the complex geometry of a portion of a bike frame.

To begin, start a new part in SolidWorks with the units set to IPS and the drafting standard set to ANSI.

Start a new sketch on the Right plane that is composed of two circles 1.5 and 1-inch in diameter centered on the origin. **Extrude** the sketch from the midplane to 3-inches.

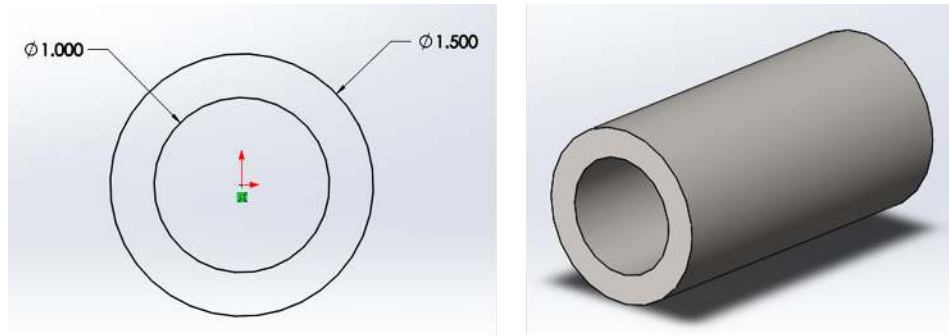


Figure 6.17 First Extruded Boss of Bike Frame

Start another new sketch on the right plane and draw a centerline vertically. Then draw two angled solid lines on both sides of the centerline from the origin, the left at 60 degrees and the right at 20 degrees. Specify the angles with the smart dimension tool by selecting the lines.

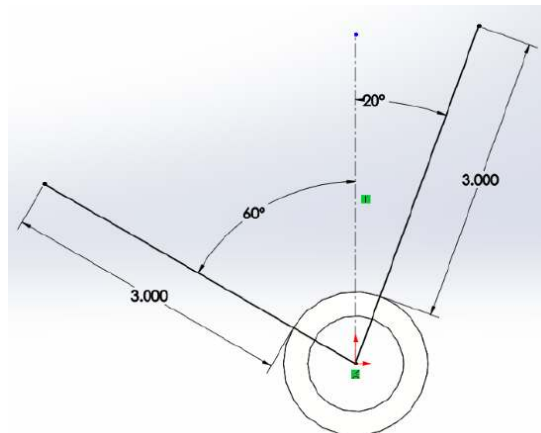


Figure 6.18 Reference Sketch for Reference Planes

Add 3-inch smart dimensions between the ends of the solid lines and the 1.5-inch circle making sure the dimensions are from the edge of the 1.5-inch circle (you can change this from the leaders' tab of the property manager of the dimension).

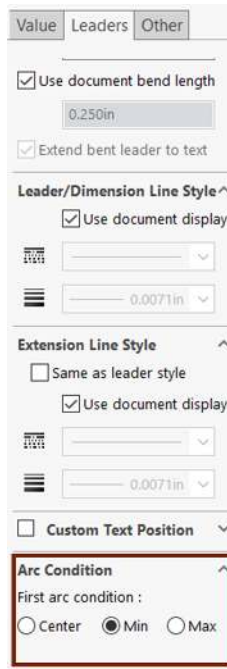


Figure 6.19 Changing Dimension Extent to Arc

Exit the sketch, **activate the reference geometry plane tool** and **choose** the end point of the right solid line from the previous sketch as well as the actual solid line. This makes the endpoint a reference for what the plane needs to go through and the solid line a reference for what the plane needs to be perpendicular to. **Confirm** the reference plane and then repeat for the left solid line.

Draw a sketch on the first reference plane you created of two circles centered on the origin with the outer diameter being 1.125-inches and the wall thickness being .125" (choose both circles with the smart dimension tool activated).

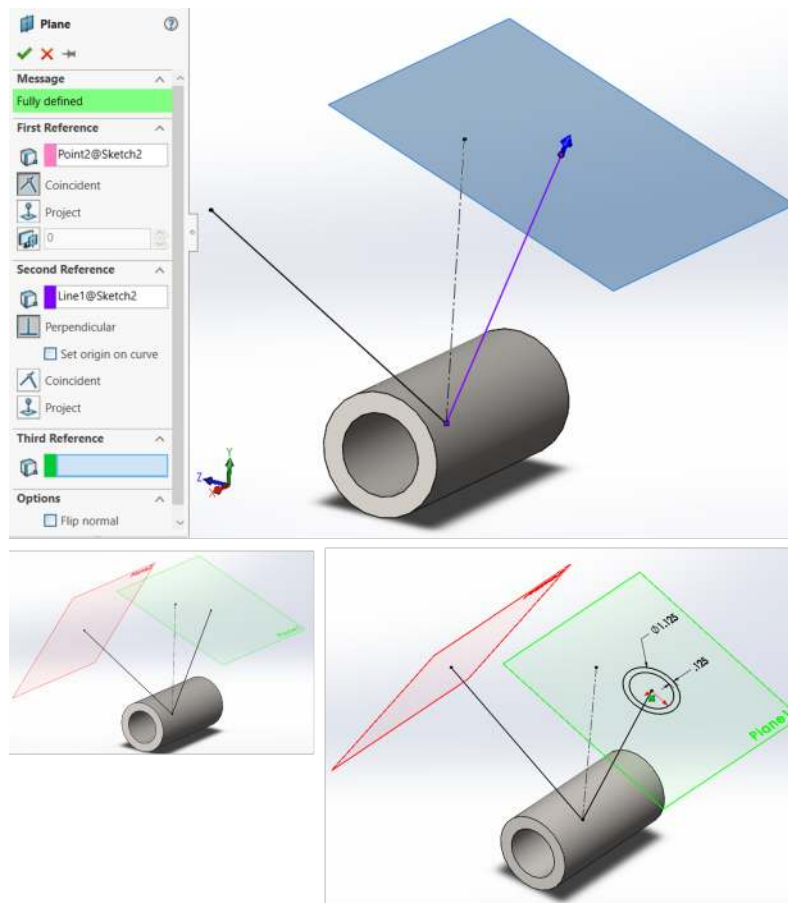


Figure 6.20 Creating Reference Plane Through End Point and Perpendicular to Line

Extrude the sketch with the “Up to Next” option selected and confirm. Repeat this process for the other sketch plane. Hide the reference planes and sketch, specify the material and save.

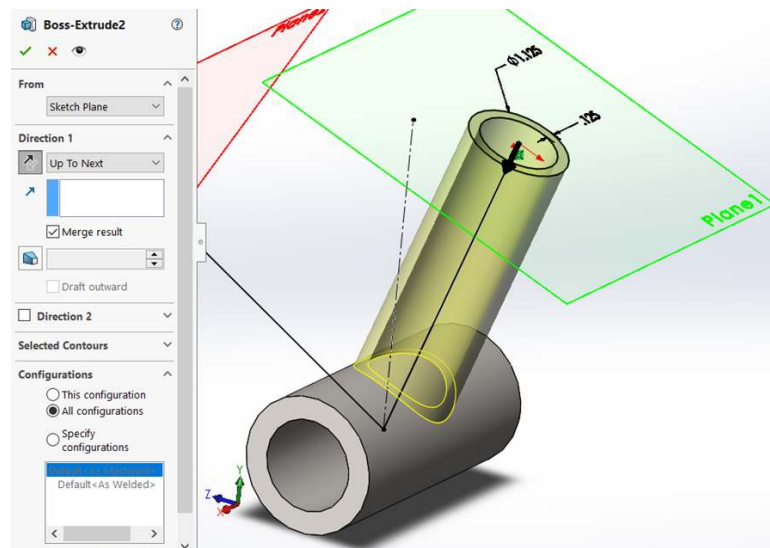


Figure 6.21 Extruding Sketch on Reference Plane

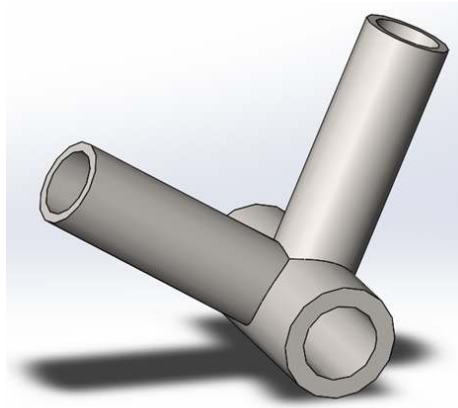
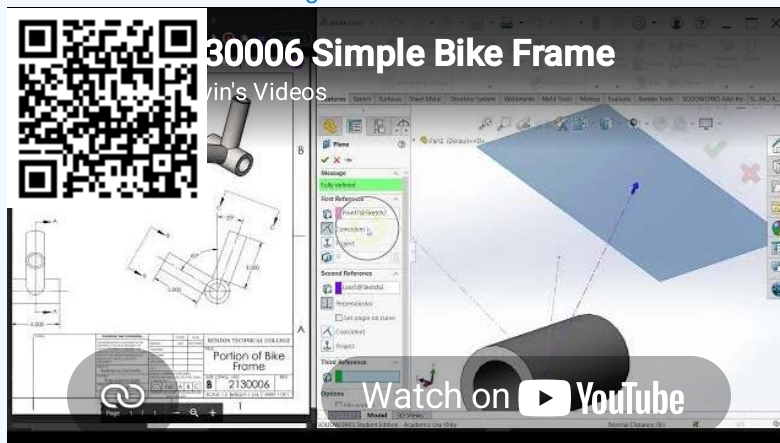


Figure 6.22 Finished Partial Bike Frame

? Exercise 6.5: Partial Bike Frame Tutorial Using Reference Planes and Reference Sketches



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6.4: Sweep

Sweep is an extrusion tool that allows you to extrude a **profile** such as a circle or a square or a complex shape along a sketched path. When you think of swept features, think of complex pipe routing or guard rails or springs. Because swept features are made up of a swept profile and a path that profile follows, they generally require two sketches (except when the profile is a simple circle or tubular shape).



Sweep has four main applications:

- Circular profile along a simple path
- Tubular profile along a simple path
- Custom profile along a simple path
- Profile along a helical path

Sweeps must follow these rules (Taken from [SolidWorks Help on Sweep Rules](#)):

- The profile must be closed for a base or boss sweep feature; the profile may be open or closed for a surface sweep feature.
- The path may be open or closed.
- The path may be a set of sketched curves contained in one sketch, a curve, or a set of model edges.
- The path must intersect the plane of the profile.
- Neither the section, the path, nor the resulting solid can be self-intersecting.
- The guide curve must be coincident with the profile or with a point in the profile sketch.

CIRCULAR PROFILE ALONG SIMPLE PATH

Doing a sweep with a circular path along a simple path is straightforward because circular profiles are built into the sweep tool. This makes it so this kind of feature only requires one sketch to create a new feature and doesn't require any sketches when only modifying a feature (see [SolidWorks Help: Creating Rods and Tubes with a Circular Profile](#)). This type of feature would be common for parts made from wire, such as springs and paper clips, or solid cylindrical bar stock that is bent, such as rebar configurations.

To do this type of sweep, first sketch your path on the plane that will get you your desired orientation. A paper clip path sketch is shown above in figure 6.23 (construction lines were added to dimension the sketch based on the wire diameter of the steel used for the paper clip). Then, you must exit the sketch and choose the sweep option. In the property manager for sweep, select the **“Circular Profile”** option and click your path sketch for the path selection box. Finally set your size and confirm. If you expect your sweep to be all one piece, such as with the paper clip, then select the **“Merge Tangent Faces”** option.

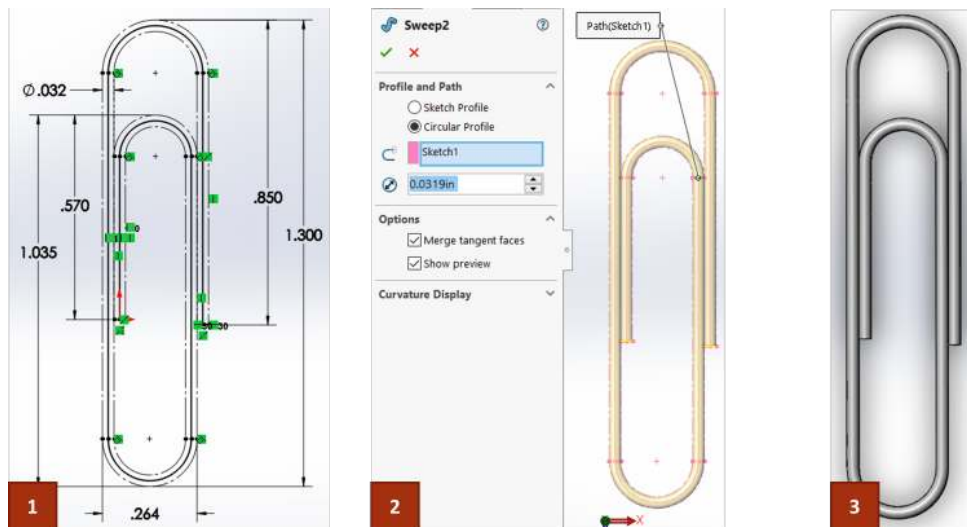
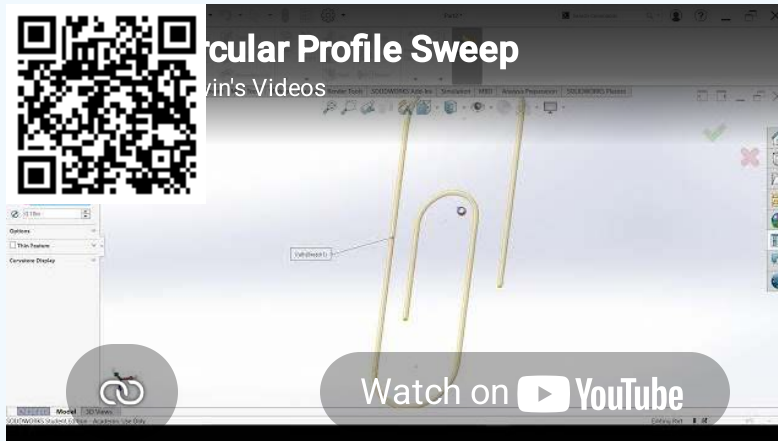


Figure 6.23 Circular Profile Sweep.: Construction lines were offset from the path in the path sketch to dimension the paper clip based on wire extents.

? Exercise 6.6 Circular Sweep

In this exercise, we will sketch a rough path sketch and sweep a circular profile along it. We won't worry about fully defining the sketch in the video for the sake of time, but you are welcome to try and reproduce the definition shown in the paper clip path sketch.

- Sketch on a default plane a rough sketch of the paper clip path using the line and arc tools.
- Activate the Sweep Boss/Base Tool and experiment with the settings to create a rough sweep of the paper clip.



TUBULAR PROFILE ALONG A SIMPLE PATH

Doing a tubular profile along a simple path is 90% like that of a circular profile except the **thin-feature** option in the sweep command is employed. We can see this through the handrail example shown below that the steps are the same as the circular profile sweep: Create a simple path sketch on the plane you desire to achieve correct orientation, exit the sketch, sweep with the circular profile and thin-feature options checked.

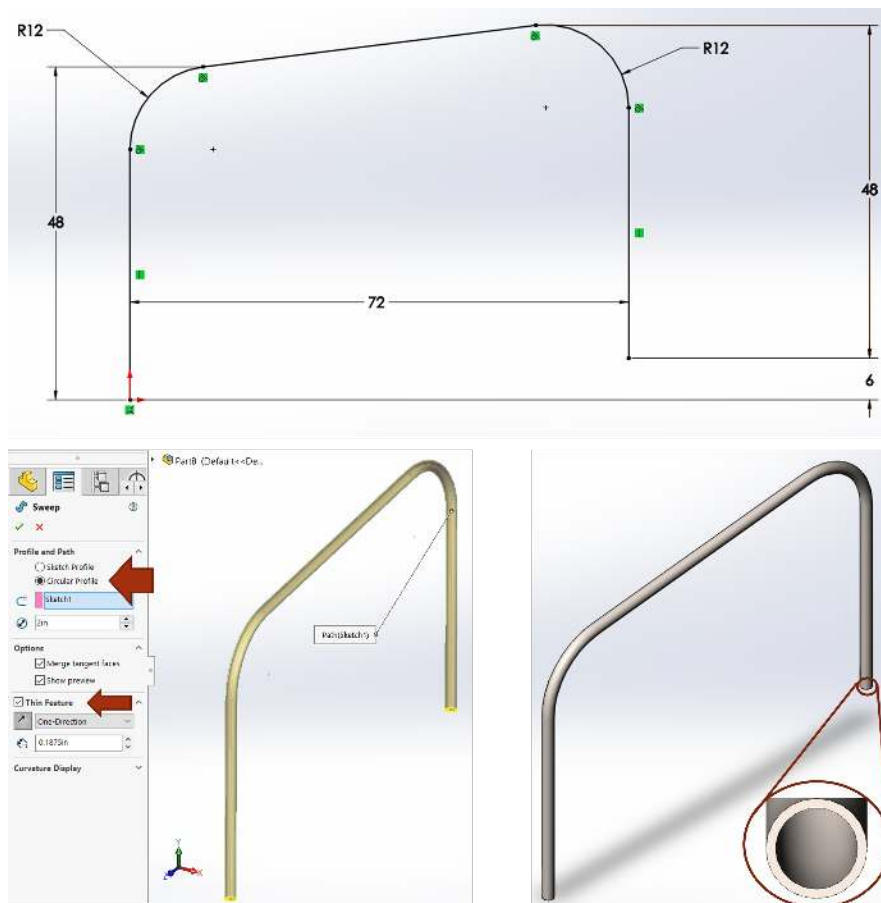
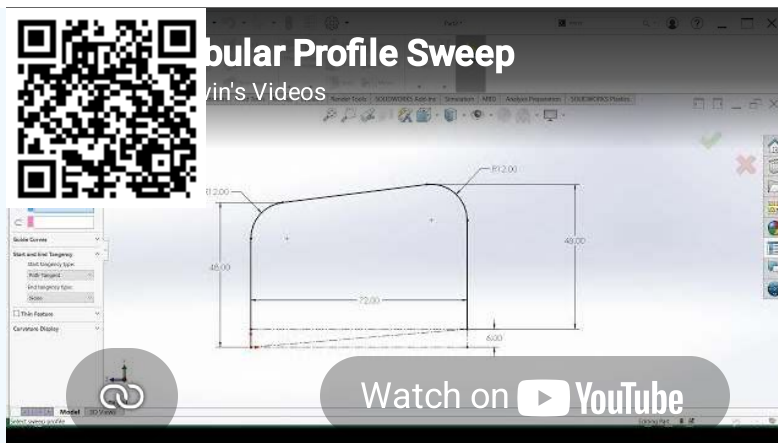


Figure 6.24 Using Circular Profile Sweep with Thin Feature

? Exercise 6.7 Tubular Sweep of Handrail

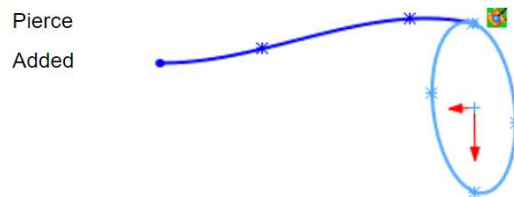
In this exercise, you will follow many of the same steps as with the circular sweep except with activating the Thin Feature option for the Swept Boss/Base to create the [2130008 Hand Rail](#).

- Open SolidWorks and start a new sketch on the Right Plane. Sketch the path of the handrail as shown in the drawing making sure that the arcs are tangent with the lines.
- Select the Sweep tool on the Features tab of the CommandManager and set the profile to circular profile and set the diameter to 2 inches.
- In the path selection box (highlighted blue) select your path sketch from the Graphics Area.
- Check the Merge Tangent Edges box and the Thin-Feature box.
- Set the Wall Thickness to .1875", zoom in on one of the ends of the handrail and make sure the wall thickness is extruding outside the sketch. If not, click the reverse direction button and confirm.
- You can extrude the end plates for this handrail as well for some extra extrude practice.
 - You should uncheck the "Merge Solids" option in your first extrude to keep the plates as separate bodies.
 - You will then need to copy the body, constraining it to the secondary location so that no information is duplicated in your modeling.
 - Ask your instructor/mentor for tips on doing this successfully and efficiently.



CUSTOM PROFILE ALONG A SIMPLE PATH

Creating a custom profile along a simple path allows you the most control over your sweep because you can make the profile any way you want and the path. However, there are more steps to doing this kind of sweep. This sweep requires two sketches: a profile sketch and a path sketch of which must be intersecting the profile sketch. You can make sure of this intersection using a new relation called the **Pierce Relation**.



Pierce relation added between sketch point and vertical line.

Figure 6.25 Pierce Relation Added (taken from SolidWorks Help)

The process of creating a swept feature with a custom profile sketch and a path sketch is first creating the path sketch and then exiting the sketch. Then you make your profile sketch making sure to add a pierce relation between the relevant line on the path sketch and the appropriate point on your profile sketch. Then you choose the Swept Boss/Base tool from the CommandManager, select your profile sketch for the profile selection box, your path sketch for the path section box, set up your settings and confirm.

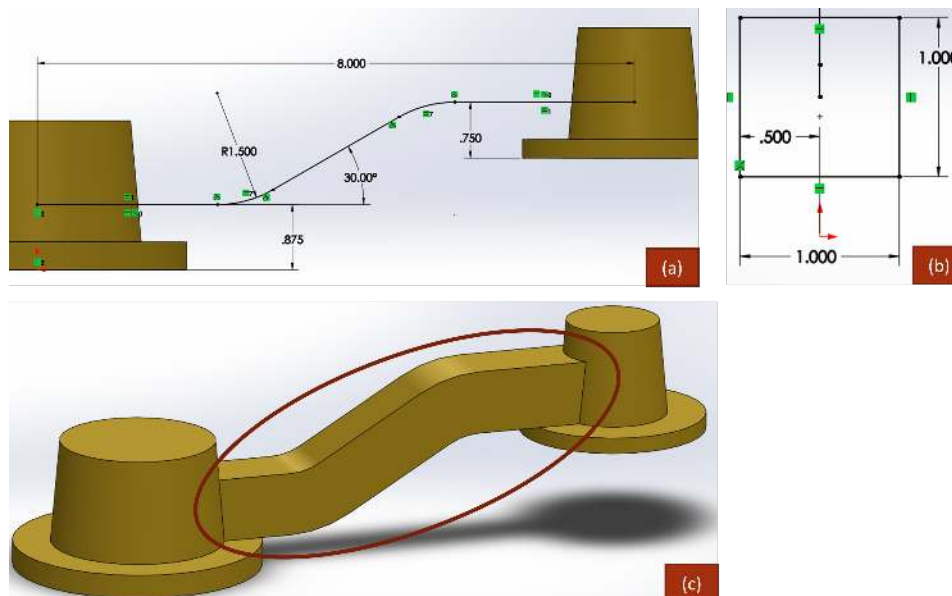


Figure 6.26 Custom Profile Sweep: (a) Path Sketch; (b) Profile Sketch; (c) Swept Feature.

? Exercise 6.8 Custom Profile Sweep

In this exercise, you will create two separate sketches: one for the profile and one for the path and use the Swept Boss/Base command to sweep the profile along the path.

- Create a rough path sketch on the right plane.
- Create a rough profile sketch on the front plane and be sure to add the pierce relation between one of the points in the profile and the path.
- Use the Swept Boss/Base tool to sweep the profile along the path.
- Note that you can also use thin feature with this kind of sweep.



PROFILE SWEEP ALONG A HELICAL PATH

A helix is a spiral in a 2 dimensional direction (see [SolidWorks Help: Helix](#) for more information). Examples of helices in action are springs, threads, slinks, coils, oil grooves etc. You as a mechanical designer will most of the time use helices for threads and springs. Within SolidWorks, there is a helix tool that allows you to automatically generate a 3D sketch of a helix for use in swept boss/bases (like with springs and coils) or swept cuts (like threads).

Figure 6.27 Curves Options Where Helix is Located

The helix and Spiral tool is located under the Curves dropdown on the features tab of the CommandManager. To create a helix, you first need a circular sketch, so the first step to creating the helix is starting a sketch on a plane that will represent the end of the helix. Then, you can go straight to the helix command to set your parameters for the helix.

When your helix is finished, if your profile is just circular, you go straight into the sweep command and proceed as you would for a circular profile along a simple path to complete your swept feature. If you require a custom path, you would create the path sketch, exit the sketch, and then proceed as you would for a custom profile simple path sweep.

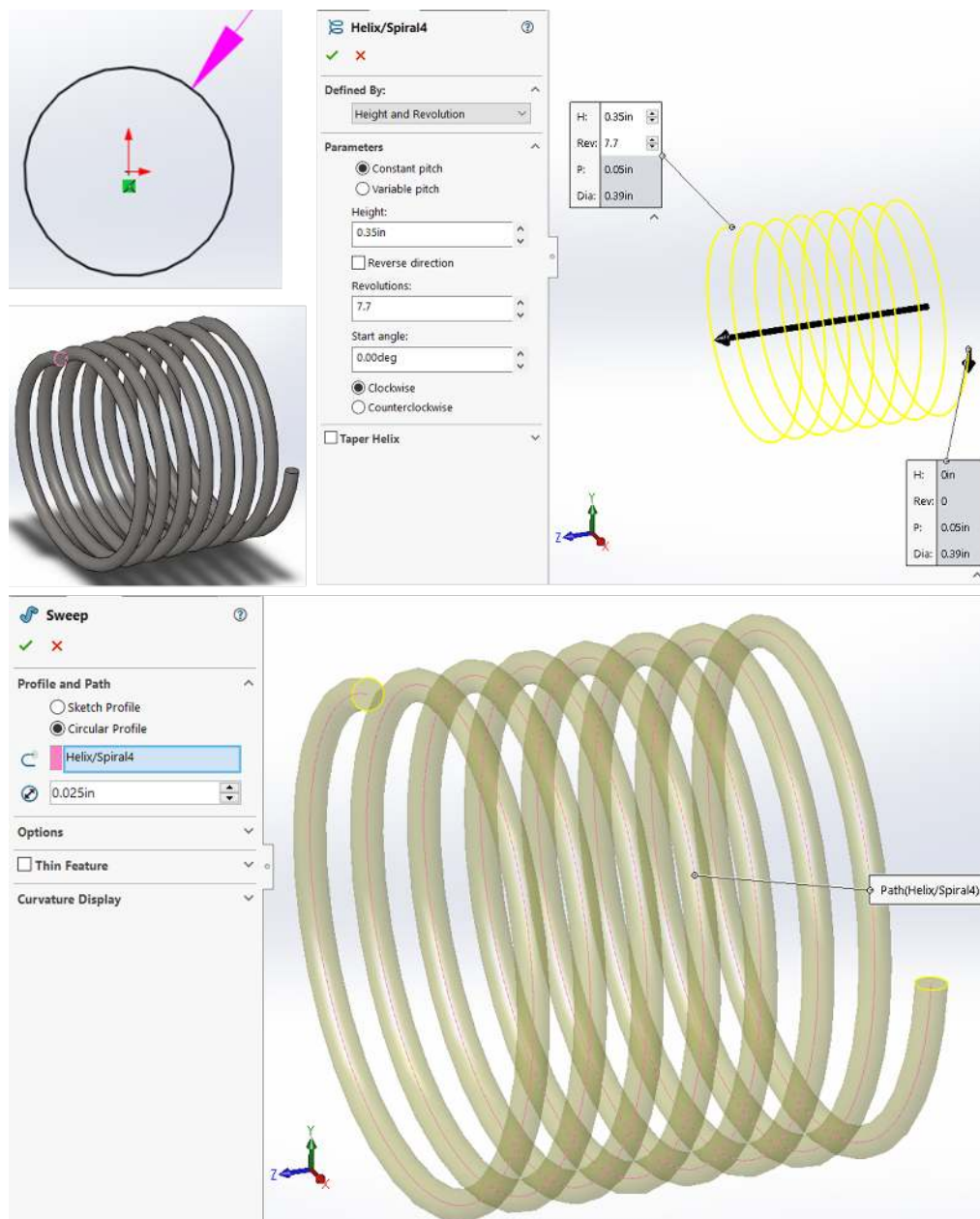


Figure 6.28 Creating a Spring with Helix and Sweep

? Exercise 6.9 Helical Sweep

In this tutorial, we will sweep a circular profile along a helical path using the helix creation tool to create [CAE1008](#), an open-ground compression spring. Video tutorials are also available to accompany this written tutorial.

CREATE THE HELICAL SPRING

- Open SolidWorks, **start a sketch** on the front plane, and sketch a .3875 inch circle at the origin.
- Activate the **Helix and Spiral** command from the drop down of **Curves** from the **CommandManager**.
- **Change** the “**Defined By**” section to “**Height and Revolutions**”, set the **height** to .35”, the **revolutions** to 7.7, and the **start angle** to 0 degrees. Also, make sure that the **rotation** is set to Clockwise.
- **Confirm the Helix/Spiral** and go straight to the **Swept Boss/Base** tool on the Features tab of the CommandManager. Set the sweep to circular profile and click on the helix for the path. **Change the diameter** of the profile to .025 inches.

GROUND OFF ENDS WITH “CUT WITH SURFACE”

- Perform a “**Cut with Surface**” using the Front Plane to make the ground geometry on the bottom of the spring.
- **Activate** the reference point command from the reference geometry drop down on the features tab of the CommandManager.
- **Click** the end face of the end of the spring on the other side of the spring and **confirm** the reference point.
- Activate the reference plane tool from the reference geometry area and choose the front plane and the reference point as references. You may have to choose these from the Feature Manager flyout if you cannot see them from the Graphics Area.
- **Confirm** the reference plane, cut with surface again to ground off the end of this side of the spring just as was done on the first end. Set the material and save the part.



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6.5: Loft and Wrap

LOFT

Loft is a tool that can be used to create some complex curvatures and shapes without reverting to surface modeling. Some example designs are heat shields, exhaust manifolds, vacuum and hair dryer attachments. We won't have an extensive section on this feature (this chapter is long enough!) but will link to some quality tutorials created by SolidWorks now that you have become familiar with the software.

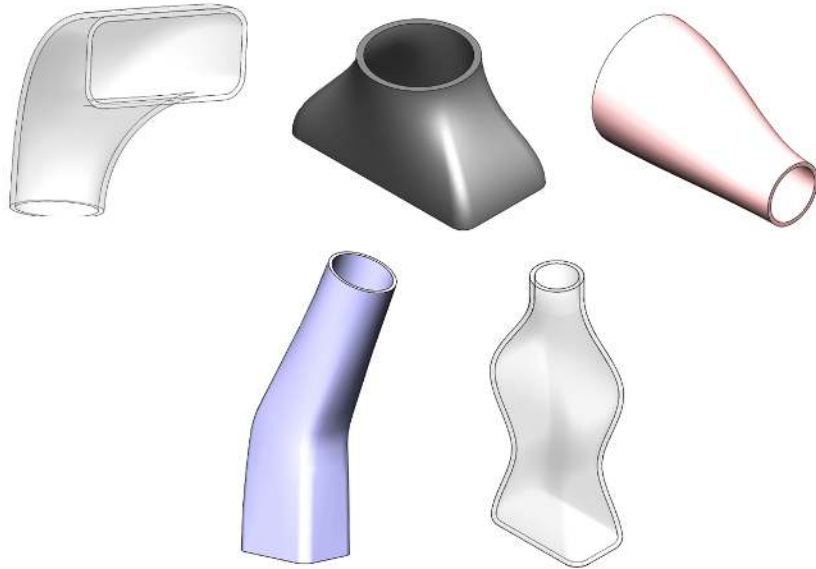


Figure 6.X Loft Examples

Some of the geometry terminology is still pretty advanced, so if you choose to learn this tool, be sure to go through the lesson with lots of pausing and looking up definitions. It is also recommended that you use the provided downloadable files in the lesson to follow along.

[SolidWorks Loft Lesson](#)

Note that you may have to sign in/create an account with MySolidWorks to access the tutorial. Use your school's or your student version serial number of SolidWorks to sign up if you haven't already.

WRAP

Wrap is another tool that allows you to create some complex geometry. Wrap has you create a sketch and then it is wrapped around a feature/body that you've already created. Wrapped cuts are most common for making parts such as cylindrical cams but wrapped bosses can be created as well for ribs on curved surfaces. Just like Loft, if you are interested, you can access a SolidWorks Lesson on using the wrap feature.

[SolidWorks Wrap Tutorial](#)

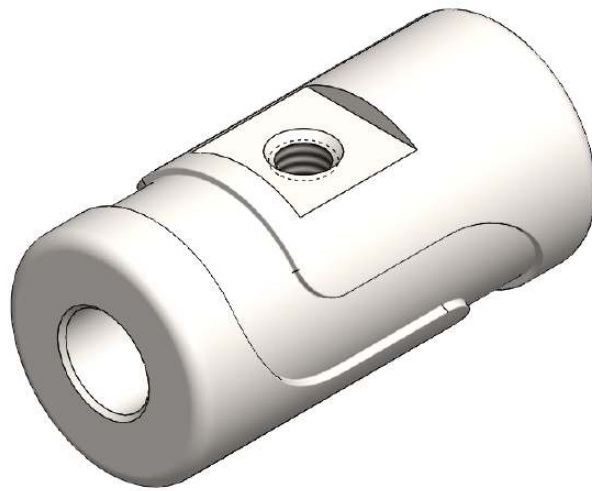


Figure 6.X Wrapped Cut on Cylindrical Cam for Sewing Machine

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CHAPTER OVERVIEW

7: Assembly Modeling

Learning Objectives

The purpose of this chapter is to...

- Model fully defined assemblies using standard and advanced mates.
- Use interference and collision detection to identify potential problems in an assembly.
- Use exploded views, Bill of Materials, and other notes to document assembly geometric and non-geometric metadata.
- Consolidate design files into a single data package that can be used as the design authority for a mechanical product.

Up to this point, we have discussed creating SolidWorks parts that can then be used to create assemblies. Assemblies are just that: parts that are put together in a specific way. The world's mechanical products are made up of assemblies. Cars, boats, planes, construction equipment, game systems; these are all made up of assemblies. In this chapter, we will learn how to not only model assemblies from SolidWorks parts, but also explore how to evaluate and document assembly designs.

[7.1: Introduction to Assembly Models](#)

[7.2: Inserting Components Into Assemblies](#)

[7.3: Standard Mates](#)

[7.4: Advanced and Mechanical Mates](#)

[7.5: Fasteners and Design Library](#)

[7.6: CAE1000 Assembly Tutorial](#)

[7.7: Assembly Evaluation](#)

[7.8: Assembly Documentation](#)

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7.1: Introduction to Assembly Models

Assembly models are just another type of SolidWorks file. You can start a new assembly file when you start SolidWorks from the Welcome Dialog box or start a new assembly from an open part, just as it is with SolidWorks drawings, by going to the New drop down on the standard toolbar or menu bar and selecting “Start New Assembly from Part”.

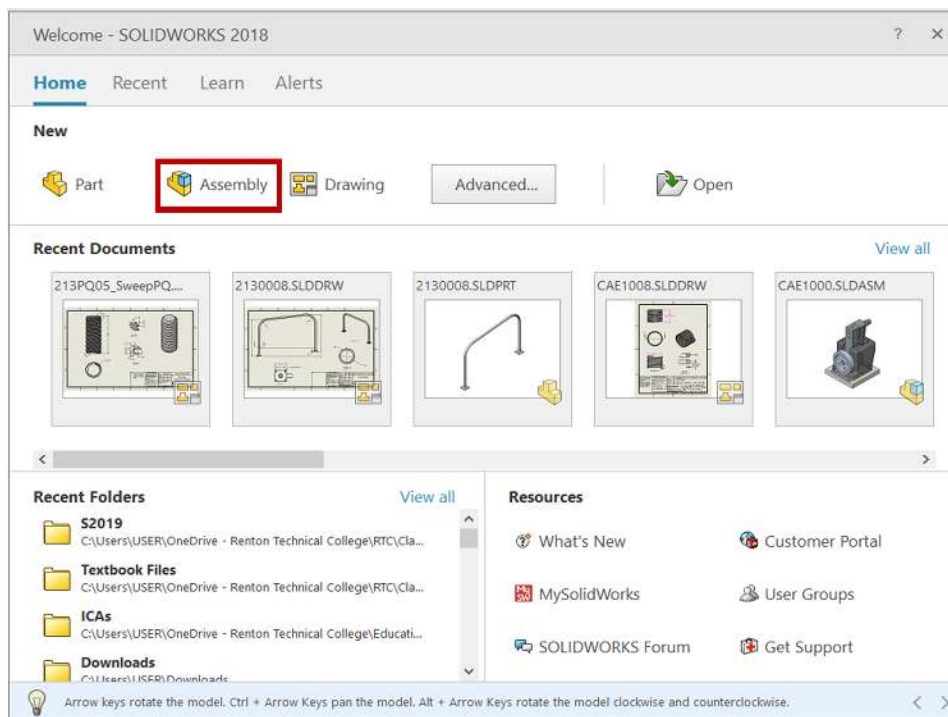


Figure 7.1 Starting a New Assembly from the Welcome Dialog Box

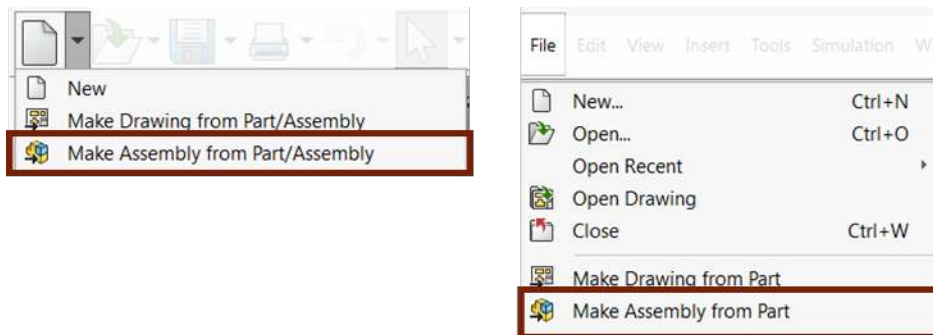


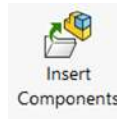
Figure 7.2 Starting an Assembly from Existing Part

There are a couple things to note about the User Interface that are different in SolidWorks assembly files. You still have the CommandManager, FeatureManager, and Property Manager. They all work the same as they do in parts. However, there are different tabs and commands in the CommandManager that are more relevant to assemblies. In the FeatureManager Design Tree, you will see a list of the parts in your assembly, and all their feature manager design trees, as well as a Matesfolder showing all of the mates in the assembly.

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7.2: Inserting Components Into Assemblies

To insert parts into an assembly, it is like moving files into a folder: you drag and drop straight from file folders or from using the Insert Component Command in the CommandManager on the Assembly tab. With this command, you find parts in your folders and then insert them by clicking to place them.



Insert Components

When you start a new assembly directly from a part, the part is put at your cursor for you to place aligned with the assembly origin. To place a part on the assembly origin, simply confirm the Insert Components Property Manager without clicking within the Graphics Area. You can also show the assembly origin and snap your cursor to the origin in the Graphics Area.

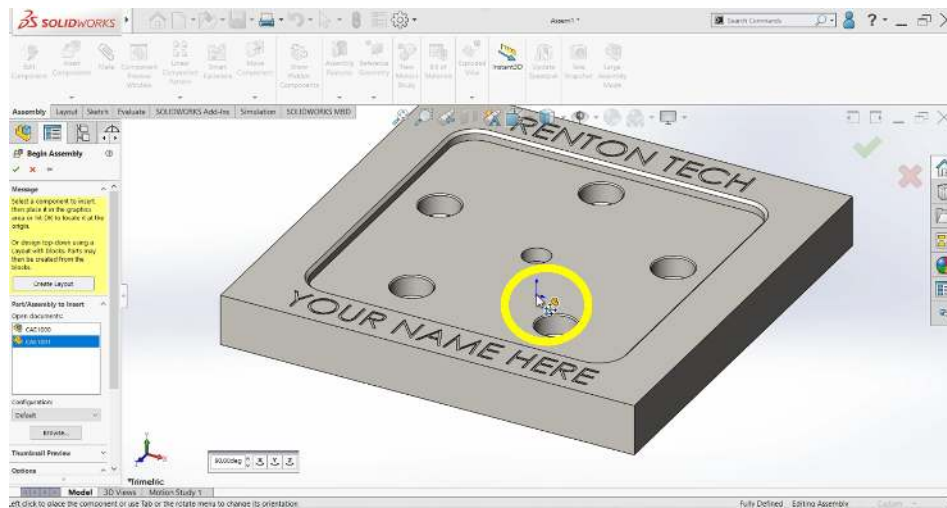


Figure 7.3 Placing First Part Origin on Assembly Origin

This is very important for design intent. If you know or are going to learn anything about GD&T, basing designs off their intended design origin is important for design changes, tolerances, and avoiding assembly errors as well. Origin placement is also key in getting center of mass measurements correct for mechanical design and for succeeding on the CSWA exam.

When you start a new assembly from scratch, you must be sure to line the first part's origin and default planes up with the assemblies. This is also important for the CSWA Exam.

? Exercise 7.1: Inserting a Component into an Assembly

For this exercise, we will start modeling the CAE1000 assembly using the parts we have previously created. To do this exercise and the future exercises in this chapter of the text, you will need all the [CAE1000 parts](#) which are linked here in a zip folder for convenience.

- Start a new assembly in SolidWorks.
- Navigate to insert a new component into the empty assembly and insert CAE1001.
- Align the part's origin with the assembly origin by confirming the Insert Components command without clicking in the graphics area.



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7.3: Standard Mates



Mate Icon

So how are parts “put together”? Well they fasten together, go into each other, become flush with each other, are pressed into each other, are allowed movement in specific directions, are made to touch in a specific way. There are all kinds of ways in which we put parts together, but how do we represent that in SolidWorks? The answer is through geometric constraints called **Mates**.

Parts within a SolidWorks assembly are constrained to one another through mates which are very similar to geometric sketch relations used in the sketch environment: coincident, concentric, tangent, distance dimension, angular dimension, etc. The most common mates in SolidWorks assemblies are **coincident** and **concentric** due to how often things are made to be flush or aligned with one another.

Mates are a command where you can add different mates to different part features such as faces, planes, edges, vertexes, axes, etc. The most robust way to apply mates is by applying them to default reference geometry, such as default planes or axes, that are linked to the design’s **datum reference frame** (a GD&T term). For more information on Mate Best Practices, see [SolidWorks Help: Mate Best Practices](#) linked here.

To apply mates, activate the mate command from the Assembly tab of the CommandManager, click on the features you want to add a mate between, and then verify your mate type. SolidWorks is usually great at knowing what kind of mate you want to add based on what you select so it will usually already be selected.

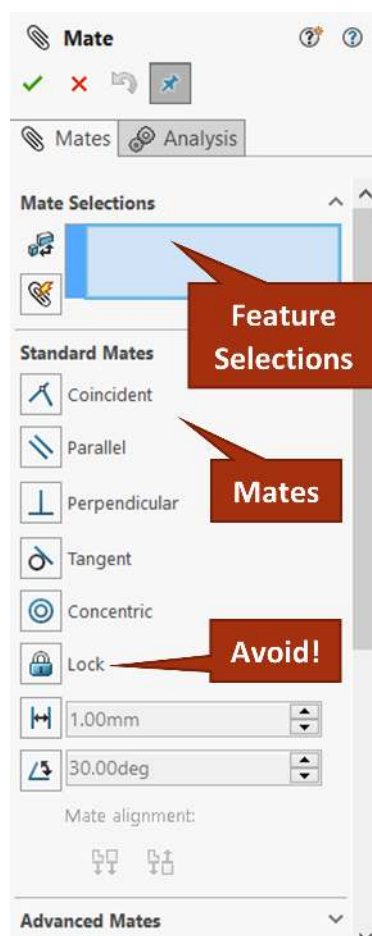


Figure 7.4 Mates Property Manager

Another way to add mates more efficiently is through **SmartMates** which is more automatic. When moving a component from the feature you want to mate, hold down the ALT key on your keyboard and move your cursor to the mating part's feature. When you release the mouse, you can verify and select the mate you desire if it wasn't already selected.

Standard mates will be the ones you use most often and are shown by default when you activate the Mates Command. They appear in the Property Manager under the heading "Standard Mates".

COINCIDENT

The **coincidentmate** is for bringing features, particularly faces/planes, together. When you want two or more faces or planes to be flush or lines up, use this mate. Select the two features you want to line up and coincident should automatically be activated. If needed, you can flip the alignment at the bottom of the Standard Mates section to make the alignment the correct orientation. When you activate different mates, a context toolbar also appears for verifying the mate type and flipping the alignment. This is to reduce mouse movement, so you don't have to move your cursor all the way to the Property Manager.

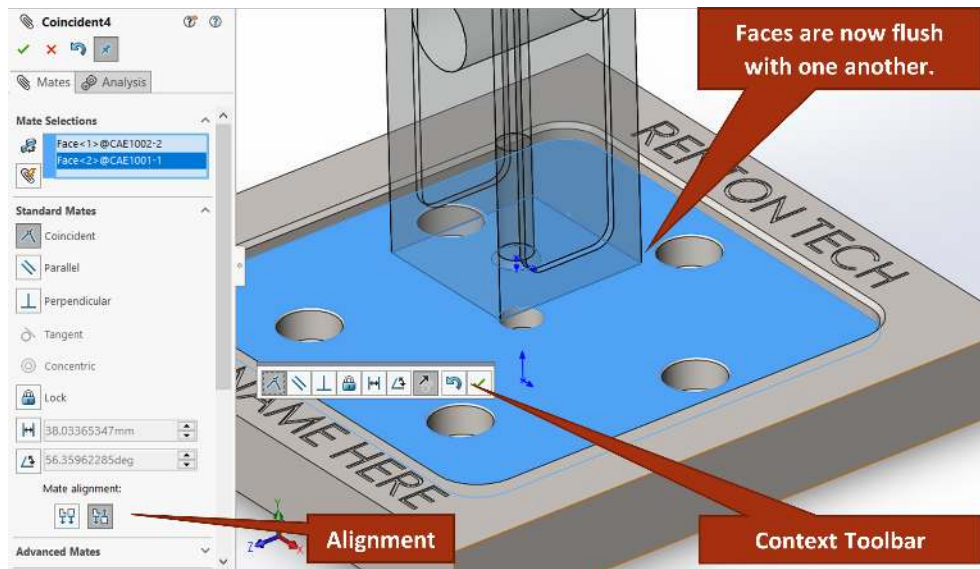


Figure 7.5 Applying a Coincident Mate to Two Faces

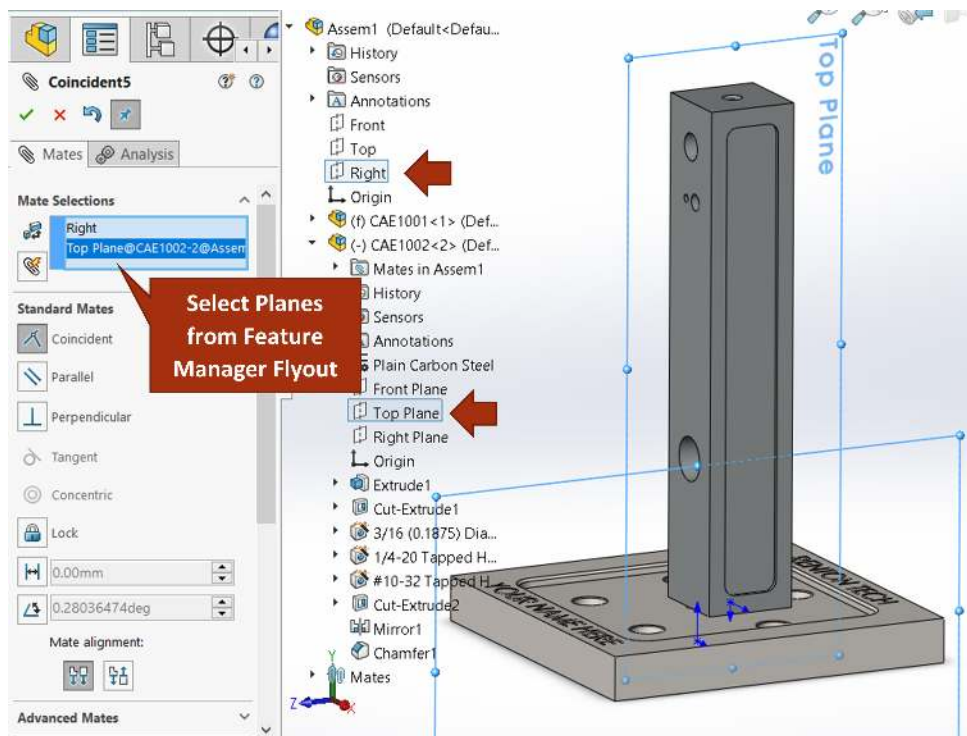


Figure 7.6 Coincident Mate Between Two Planes

? Exercise 7.2 Using Coincident Mates

In this exercise, we will continue with the CAE1000 assembly and add many of the coincident mates that are in the assembly for CAE1001, CAE1002, and CAE1003.

- Insert CAE1002 and CAE1003 into the assembly. Move and rotate parts into approximate location and orientation.
- Add coincident mates where applicable between the three parts:
 - The top face of CAE1001 and bottom face of CAE1002.
 - The Front Plane of CAE1002 and the Front Plane of the assembly.
 - The Right Plane of CAE1002 and the Right Plane of the assembly.
 - The back face of CAE1002 and the non-cut face of CAE1003.



CONCENTRIC

Concentric centers cylindrical features together: pins to holes, holes to holes, tapered shafts to tapered holes, etc. Concentric is used when the cylindrical feature is not lined up with the default planes of the part model (in terms of GD&T, it isn't a datum feature).

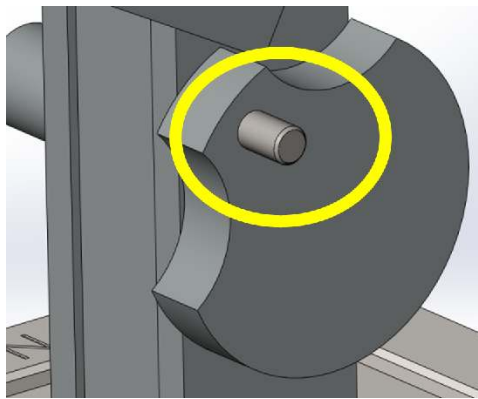


Figure 7.7 Press Fit Pin Required Lock Rotation Option in Concentric Mate

The **Lock Rotation** option is for when you are lining up press fit cylindrical features, fasteners, hubs fixed with a set screw, or other features you don't intend to be free to rotate. You can check the Lock Rotation option to prevent rotation of the mate combination to better match design intent and to give more accurate results in motion studies and analysis. For example, in figure 7.7, the pin is intended to be press fit into the crank.

Like with Coincident, you can flip the alignment of concentric mates as well to ensure the features are oriented appropriately.

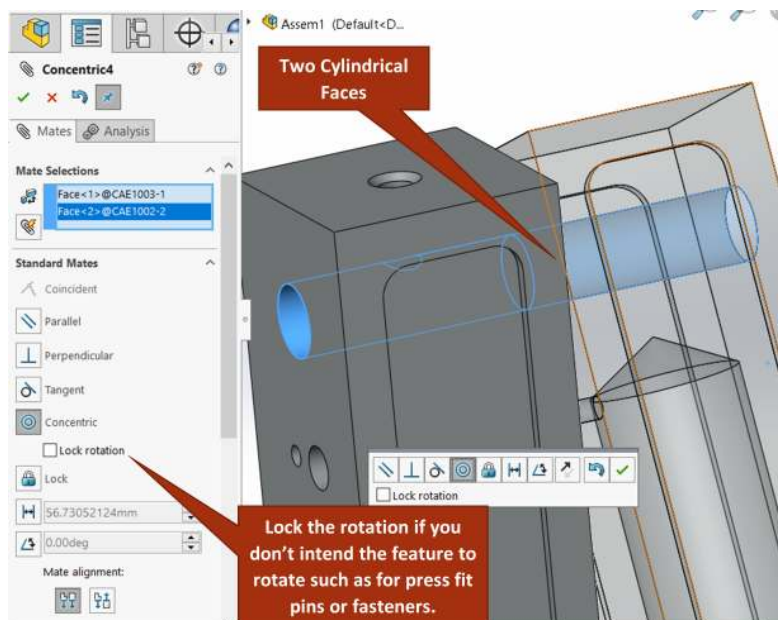


Figure 7.8 Concentric Mate Applied to Two Cylindrical Surfaces

? Exercise 7.3 Concentric Mates

Continuing to model the CAE1000 assembly, here we will add concentric mates where applicable to the rest of the parts of the assembly.

- Add a concentric mate between the top .3125" hole of CAE1003 and the top matching hole of CAE1002.
- Insert the remaining components CAE1004, CAE1005, CAE1006, CAE1007.
- Add the concentric mates as are needed between the added in parts.



PARALLEL AND PERPENDICULAR

Parallel and Perpendicular mates will make planar or cylindrical features parallel or perpendicular as you can imagine. These two mates are very seldom used. Although it is perfectly acceptable to imagine a cylindrical feature perpendicular or parallel to a planar feature, SolidWorks will not allow parallel/perpendicular mates to be applied to planar and cylindrical faces together; you must do two or more planar features or two or more cylindrical features, not mix and match.

TANGENT

Tangent mates are used most often for touching planar features to cylindrical features together or for touching **eccentric** circles together. Some examples are pin/hole combos with lots of clearance and set screw ends onto cylindrical shafts. See the example below where a set screw bottom is being connected to a shaft using a tangent mate.

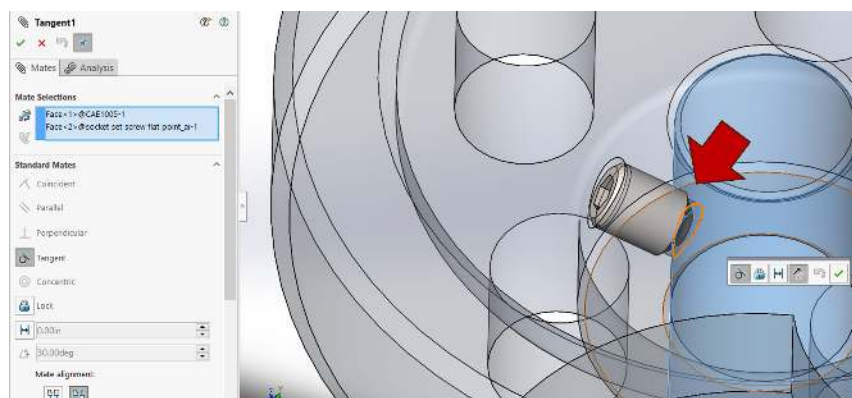


Figure 7.9 Tangent Mate on Flat End of Set Screw and Cylindrical Shaft

DISTANCE AND ANGLE

Distance mates and **Angle mates** are used to specify spaces between two features. In the figure below, you can see .030 inches being applied between two planar faces. The same can be done for angles. Distance and angular mates are popular for moving components (when a limit mate isn't necessary).

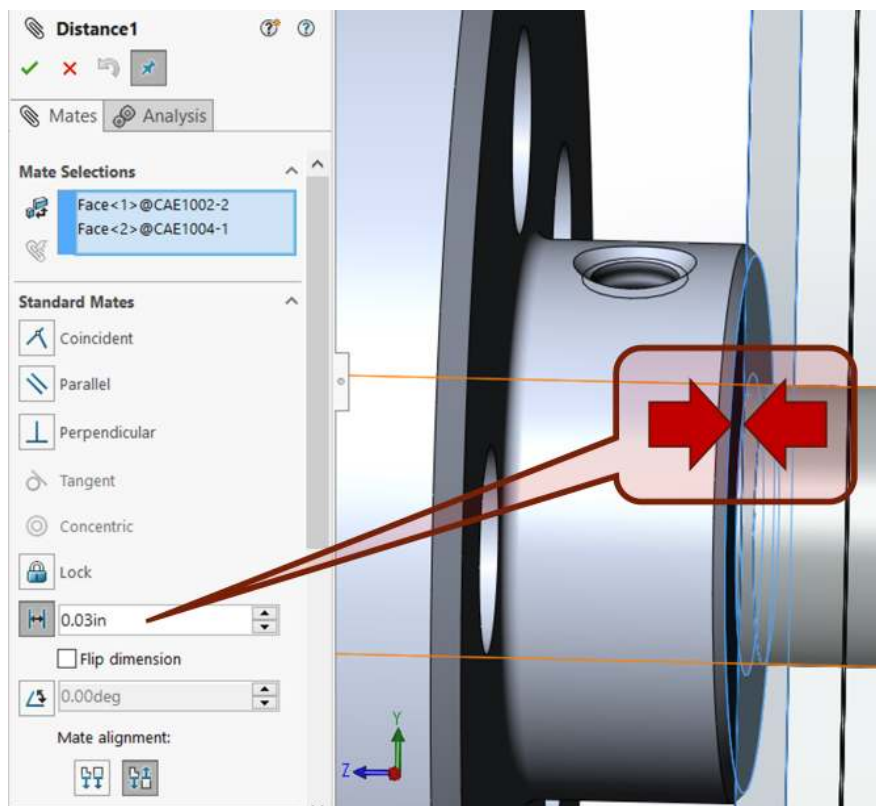
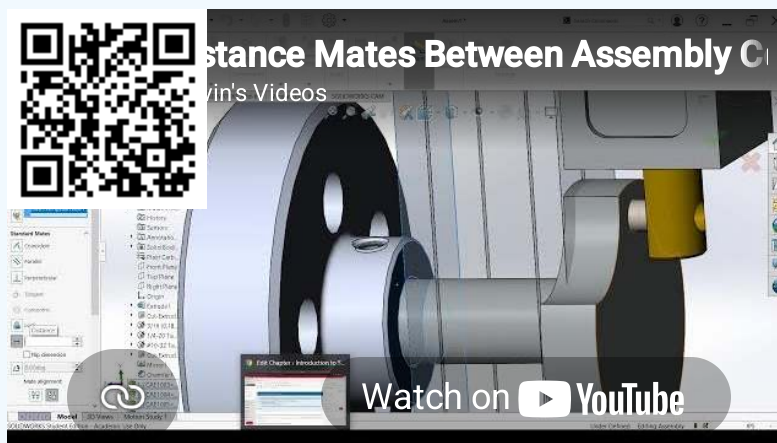


Figure 7.10 Distance Mate Between Two Planar Surfaces on Flywheel and Body

? Exercise 7.4 Distance Mate Between Planar Surfaces

In this exercise, still modeling CAE1000, you will add a distance mate between the planar face of the flywheel and the body of the compressed air engine. To add the mate, you will do the same procedure as for any other standard mate except input a distance when adding the mate that represents the amount you want the two faces spaced.

- Activate the Mate Command.
- Select the two faces to add a distance mate to.
- Choose the distance mate option from the context toolbar or the property manager, and input the distance where prompted.
- Add remaining parts and mates to the assembly with the exception of the fasteners (we will get to those).





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7.4: Advanced and Mechanical Mates

Advanced mates allow you more control and more specified constraints for constraining parts to other parts. The main ones here are **Width** and **Limit** mates. There is also symmetric, linear coupler, profile center and path mates here. They work very similar to standard mates, they are just more options and more selected entities to use them.

WIDTH

A **Width Mate** is used to center parts within other parts using planar faces. If you think about it, width does the same thing concentric does but for prismatic parts as opposed to cylindrical parts.

To use width, you select the two faces a part will be centered between and then the two faces you will use as the centering reference on the part you want to center. Use width when default planes and coincident wouldn't be appropriate (most the time we center parts in relation to the default planes so we can just use them as opposed to width).

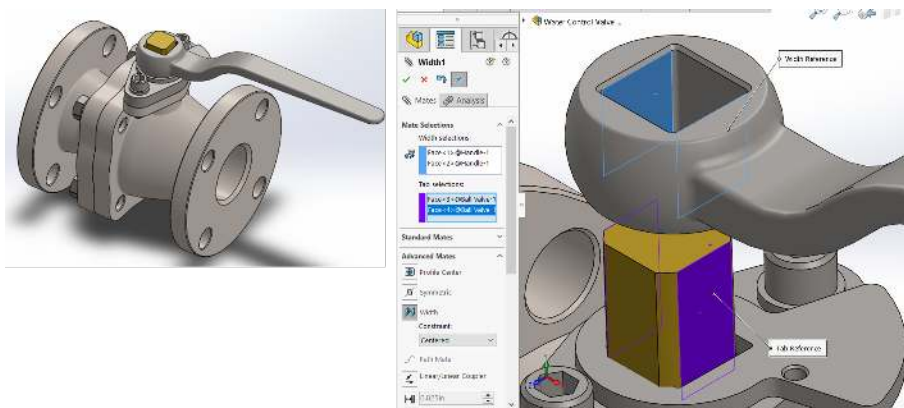


Figure 7.11 Width Mate Used to Center Handle Broach on Pipe Valve Key

LIMIT DISTANCE AND ANGLE

Limit distance and angle mates work just as the standard distance and angle except you can put in a range rather than a single value. This is great for making assemblies dynamic: dynamic hydraulic cylinders or valves or actuators. You specify your minimum and maximum distance/angle and the faces that distance/angle will be between.

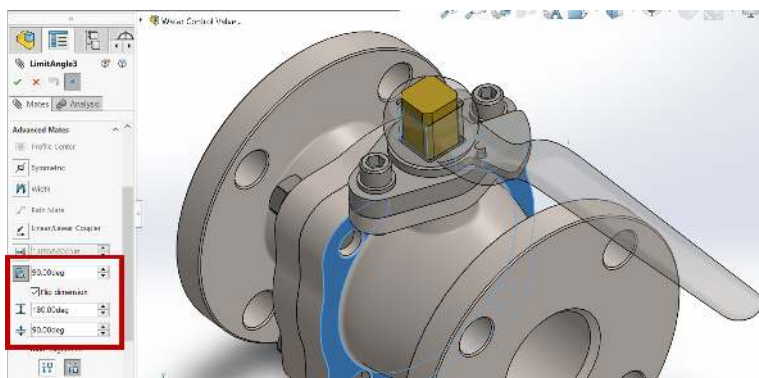


Figure 7.12 Limit Angle Mate Being Used on Handle to Limit Rotation to a 90 Degree Turn

MECHANICAL MATES

Mechanical mates are also available for more specific mechanical relationships between parts such as screw/thread mate (a part turns as it is translated along another part), gear mate (as one part turns at one speed, another part turns at a different speed), rack and pinion, cam, slot, hinge, and universal joints. We won't get into these here but you can look them up in more detail from SolidWorks Help as well as from [MySolidWorks Training on Mechanical Mates](https://workforce.libretexts.org/@go/page/31556).

? Exercise 7.5 Advanced and Mechanical Mates

In this exercise, you will be given the opportunity to explore advanced and mechanical mates through the MySolidWorks portal. Below are two training lessons that you can follow along with to explore the two mate categories.

- [MySolidWorks: Advanced Mates Training](#)
- [MySolidWorks: Mechanical Mates Training](#)

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7.5: Fasteners and Design Library

The **Design Library** is in the task pane and has a link to the SolidWorks Toolbox which is included with most versions of SolidWorks. The design library is filled with common components used in mechanical design and that are commonly sold by suppliers. The most common use of the design library is for fasteners such as screws, bolts, nuts, set screws, washers, etc.

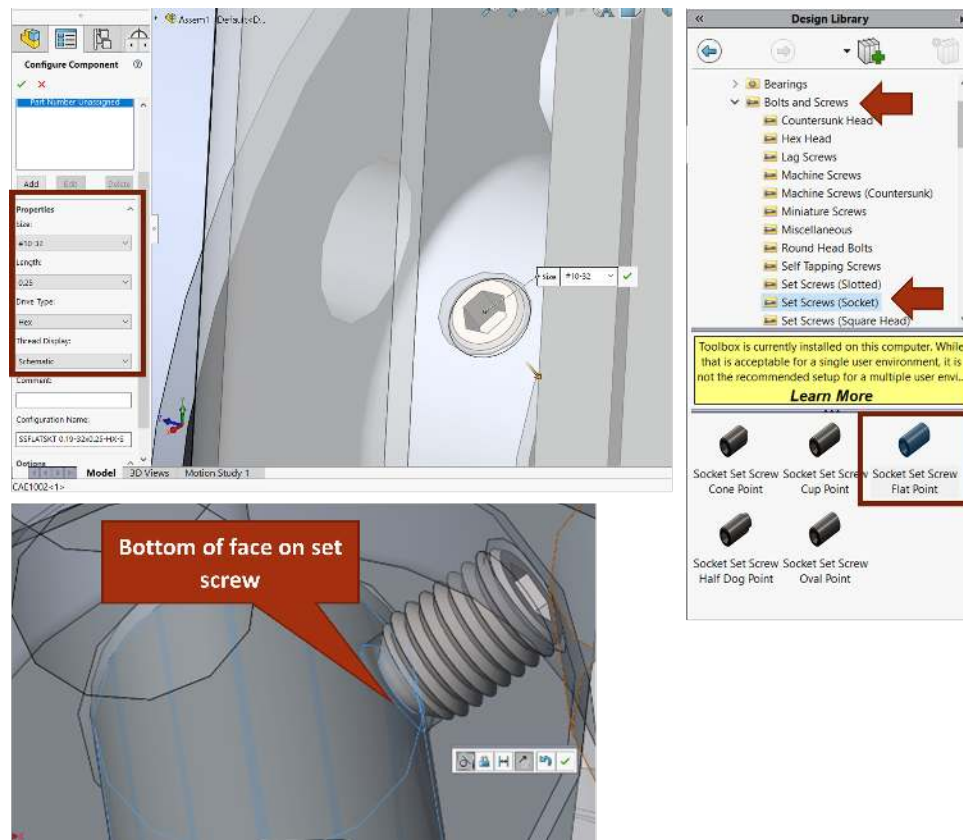


Figure 7.30 Flywheel Set Screw

SolidWorks introduced a tool called Smart Fasteners years ago to try to make the fastener specification process faster, but it still holds many bugs. It is only best to use this tool when adding more than a dozen fasteners.

To add fasteners from the SolidWorks toolbox, open the task pane, find the fastener you want to add, and drag and drop it where you want it. Automatic mates will be added if you select a compatible edge. Once you drop the fastener, you can specify all the type settings. Once confirmed, you can add more of the same type of fastener. This feature is great for common fasteners, but if you need specialty fasteners, use models from a common supplier website such as [McMaster-Carr](https://www.mcmaster.com/).

Double check your mates when placing toolbox components!

? Exercise 7.6 Specifying Fasteners in CAE1000

In this exercise, you will add a few of the many fasteners that are contained within CAE1000. The rest you will add in the upcoming tutorial. To add the fasteners, you will load the design library and drag components into place. You may also need to add applicable mates to constrain the fasteners to the assembly according to design intent.

- First Add CAE1009 with appropriate mates.
- Load the design library from the Task Pane.
- Add a 1/4-20 Socket Head Cap Screw to the 1/4 counterbore hole at the bottom of the assembly. Make sure to drag over the inset edge of the counterbore so that automatic coincident and concentric mates are added.
- Add a 5/16-18, 3-inch length and 7/8-inch threaded length hex bolt to the body.

- Add the 5/16" Plain Type A Washers and 5/16-18 Hex Nuts assembled with CAE1008.
- Also add the #10-32 by .25" length set screw to the flywheel tapped hole and make the bottom face of the set screw tangent with the crankshaft.
- #10-32 by .375-inch length set screw between CAE1002 and the hex bolt.
- Add the remaining fasteners and parts to the assembly.



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7.6: CAE1000 Assembly Tutorial

This tutorial is to provide a review of all the things we have done with CAE1000 in this chapter and to put all the instructions in one place. There are also a few alternative ways in modeling the assembly that are presented here that weren't presented in the previous video exercises of this chapter.

We modeled [CAE1000](#) as a SolidWorks assembly applying standard mates to planes and faces as well as using toolbox components in design. To model the assembly, you needed the [CAE1000 Part files](#). You should remember modeling most these parts throughout the textbook. When downloading the parts, it is crucial to remember to extract all the files before trying to use them. This is because the files are given in a compressed zip folder format.

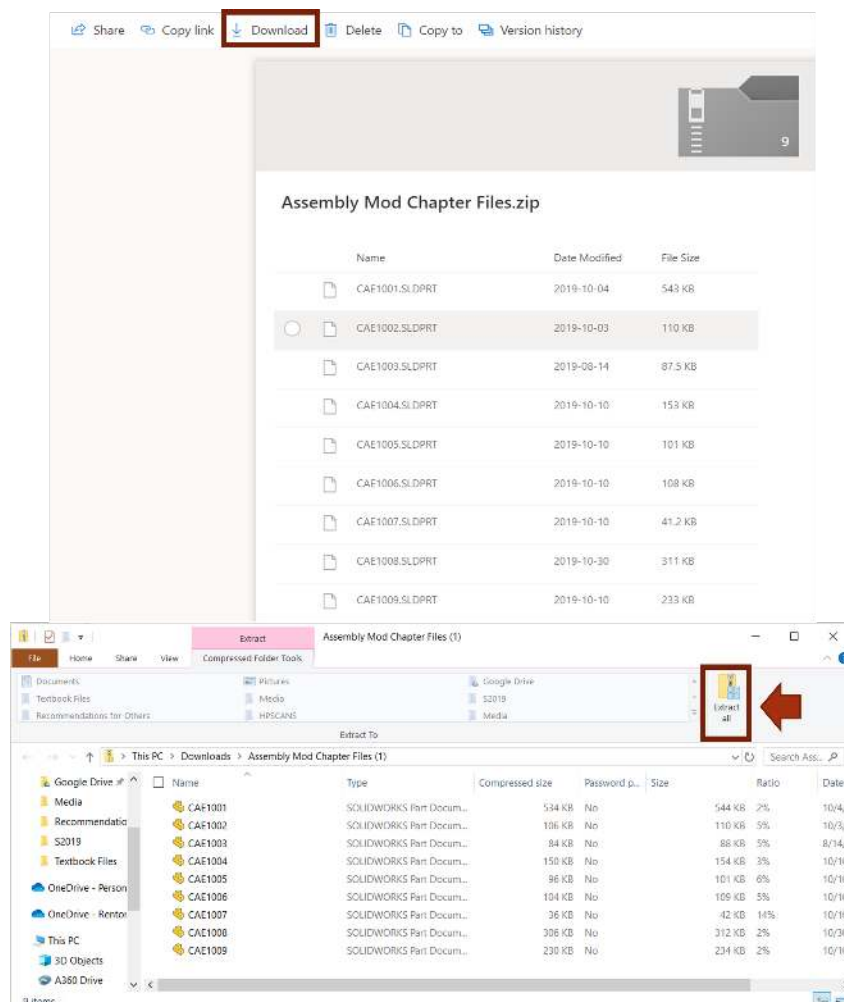


Figure 7.15 Downloading Part Files for Assembly

INSERT THE FIRST COMPONENTS

Open SolidWorks, download and extract the part files, and start a new assembly file. Check your units to make sure they are in inches.

As soon as you open the assembly, the **Insert Components** command should activate with a file folder. Navigate to the downloaded part files via the file folder. Select CAE1001 from the list, click “Open” in the dialog box, and confirm the command to align the part and assembly origins.

Activate the “Insert Components” command from the assembly tab of the CommandManager, click “browse” and select CAE1002 to insert next. Insert this part anywhere in the graphics area that is NOT the origin.

Rotate the part into the approximate assembly orientation by holding down the RIGHT mouse button and moving the mouse.

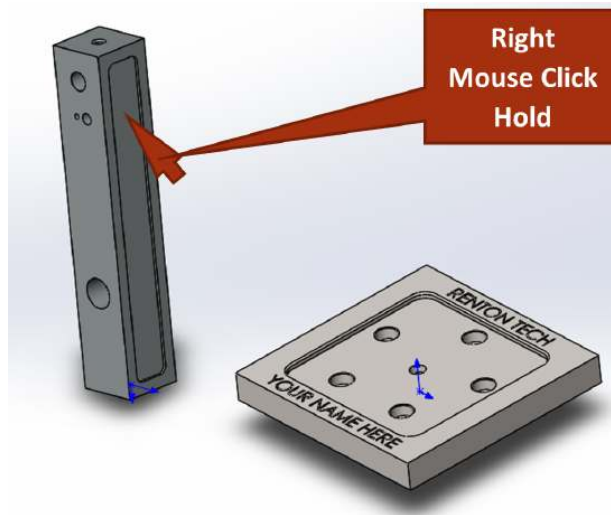


Figure 7.16 Rotate Components in Graphics Area

INITIAL COINCIDENT MATES

Activate the Mates command from the Assembly Tab of the CommandManager, click the bottom face of CAE1002 and the top internal pocket face of CAE1001. Coincident Mate should be automatic. Confirm the Mate.

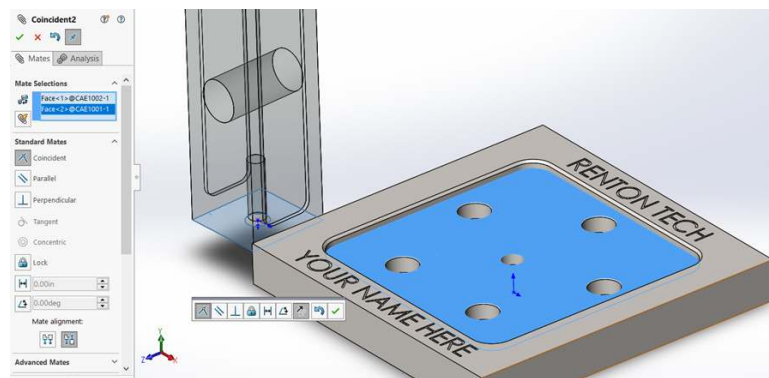


Figure 7.17 Coincident Mate Between Top and Bottom Faces

Then while still in the Mates Property Manager, open the feature manager design tree flyout, select the assembly Front Plane and CAE1002 Front Plane and the automatic mate should be coincident. Repeat this for the Assembly Right Plane and CAE1002 Top Plane. The orientation of your assembly should be as shown below.

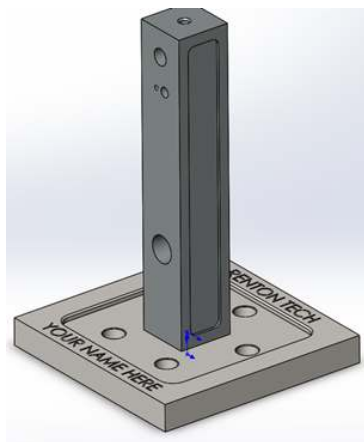


Figure 7.18 Orientation of CAE1001 and CAE1002

Activate the insert components command again from the assembly tab of the CommandManager and insert CAE1003 anywhere in the graphics area.

Add mates using the Mates Command:

- A coincident mate between the cut face of CAE1003 and the back face of CAE1002.
- A concentric mate between the top hole of both parts.

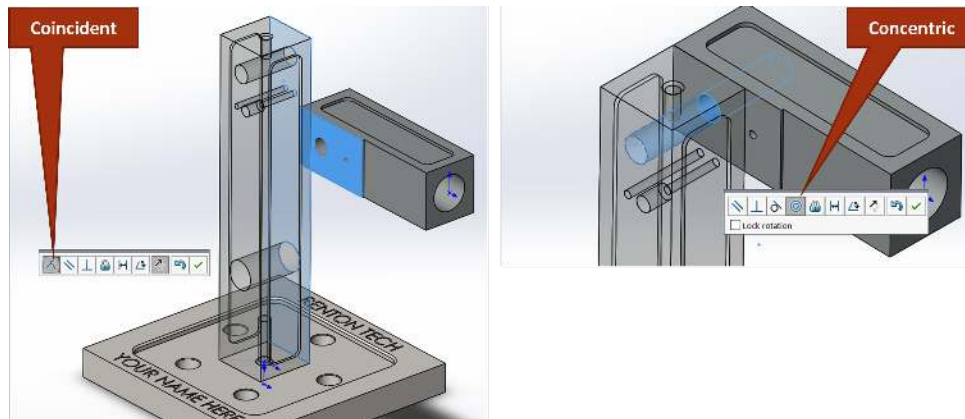


Figure 7.19 Coincident and Concentric Mates Between CAE1002 and CAE1003

MATES WITH FLYWHEEL, CRANK, PISTON, AND PIN

Insert CAE1004, CAE1005, CAE1006, and CAE1007 into the graphics area using the Insert Components Command. Also rotate them into approximate orientations with the Right Mouse Click option.

Apply mates to these three parts:

- Concentric mate between CAE1003 and CAE1004.
- Concentric mate between CAE1004 and CAE1005 using the Lock Rotation Option.
- Concentric mates between CAE1005 and CAE1006.
- Concentric mates between CAE1003 and CAE1006 using the Lock Rotation Option.
- Concentric mates between CAE1005 and CAE1007 using the Lock Rotation Option.
- Coincident mate between CAE1004 and CAE1005.
- Coincident mate between CAE1005 and CAE1007.
- A distance mate between CAE1002 and CAE1004 at .030 inches.

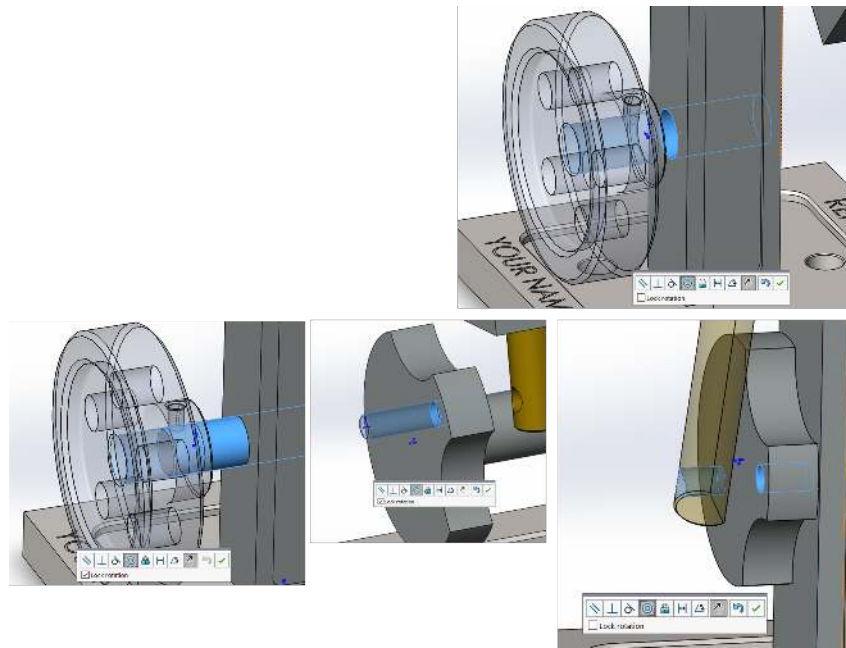


Figure 7.20 Mates with Flywheel, Crank, Piston, and Pin

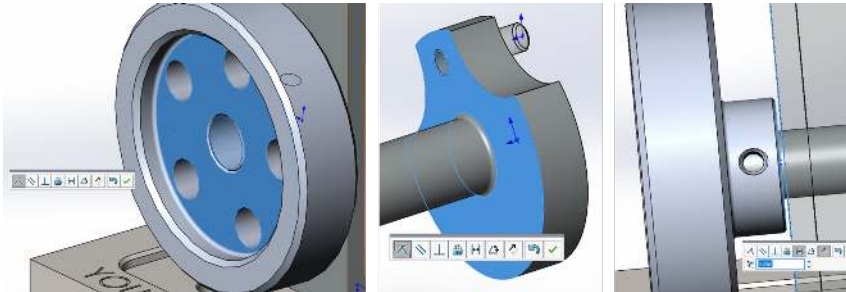


Figure 7.21 Coincident and Distance Mates for Flywheel, Crank, and Pin

CAE1009 ENCLOSURE

You can turn off the origins via the Heads-Up View Toolbar because at this point, they are becoming distracting.

The final design component to insert is CAE1009 using the Insert Components command. Rotate the component into the approximate orientation and add the following mates to CAE1009:

- Coincident mate between the Top Plane of CAE1009 and the Top Pocket Face of CAE1001.
- Coincident mate between the Front Plane of CAE1009 and the Front Plane of the Assembly.
- Coincident mate between the Right Plane of CAE1009 and the Right Plane of the Assembly.

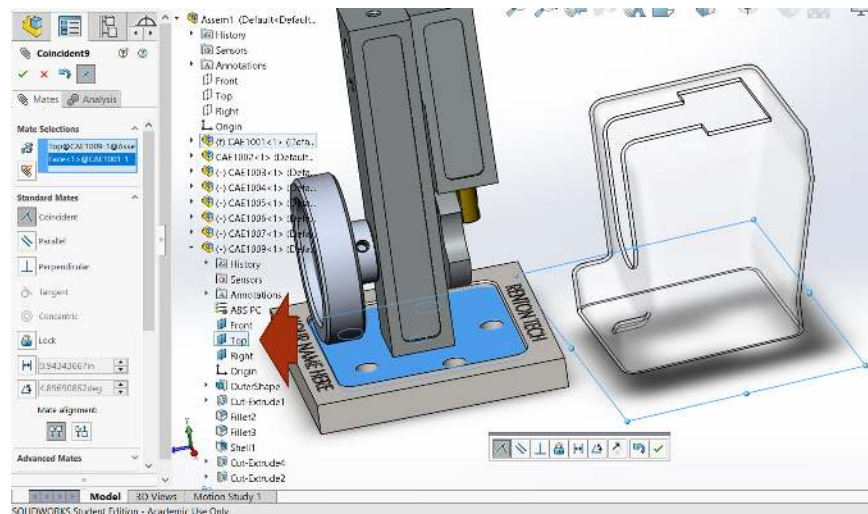


Figure 7.22 CAE1009 Coincident Mate on Bottom Face

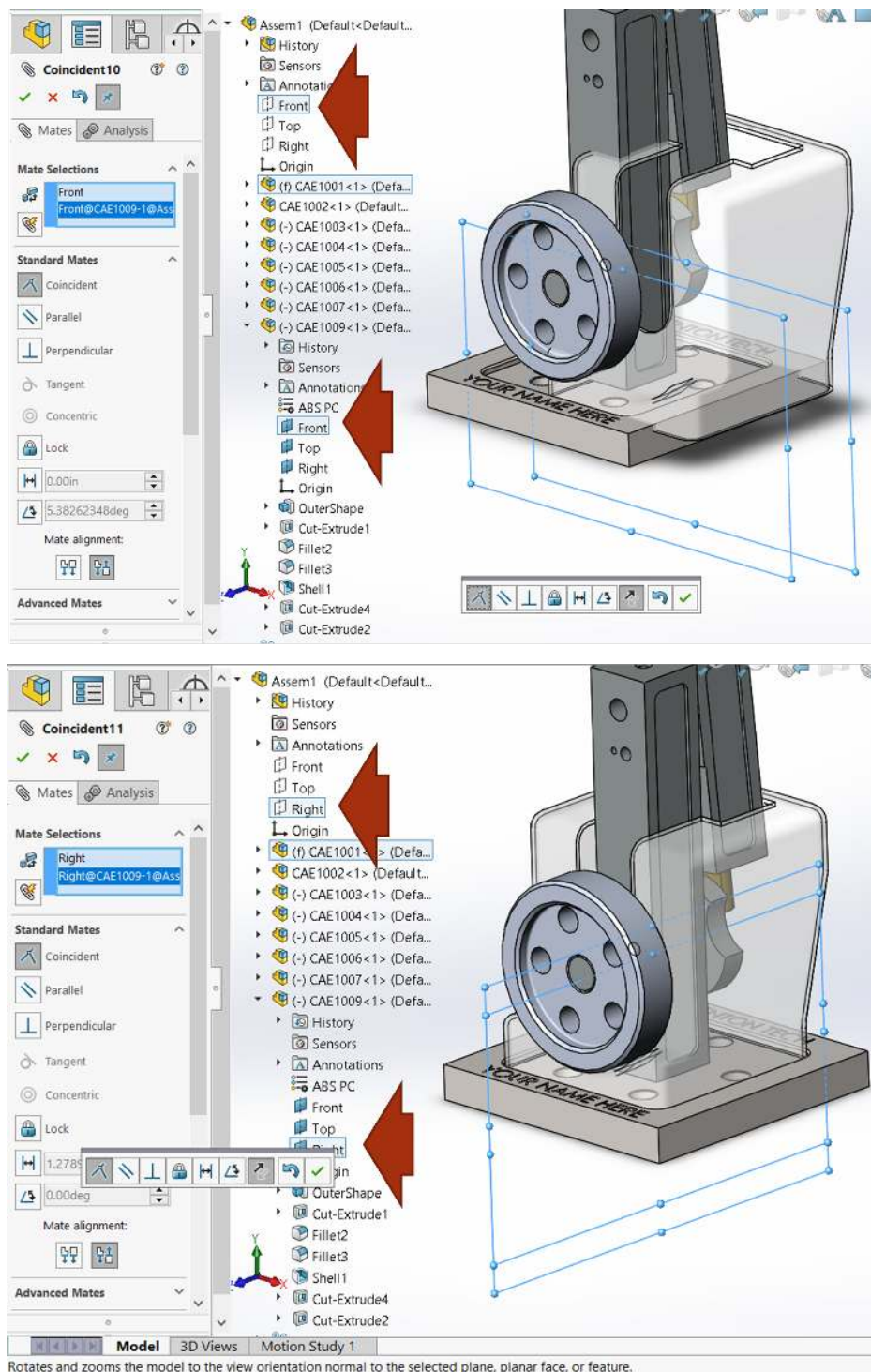


Figure 7.23 CAE1009 Plane Mates

TOOLBOX COMPONENTS

From the task pane on the right side of your screen, click on the Design Library tab. Click on the toolbox. If it is not already loaded in, then load it in.

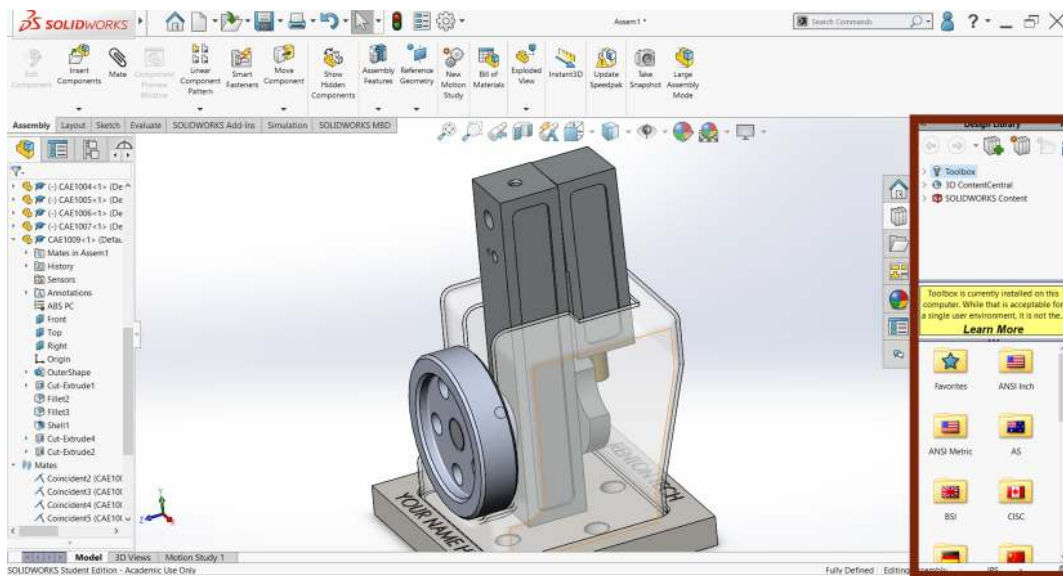


Figure 7.24 Design Library in Task Pane

Navigate to the Hex Head folder by clicking ANSI Inch, Bolts and Screws, Hex Head.

Drag and drop the hex bolt near the left side of the top hole. Set the bolt size to 5/16-18, 3-inch length and 7/8-inch threaded length. Also, choose to show the schematic version of the thread and confirm the property manager for the fastener. For inserting more components, press the red X in the property manager to confirm the command.

Dragging and dropping will sometimes add automatic mates!

Check the mates added if any. We need a concentric mate between the bolt and the left hole locking the rotation and a coincident mate between the left face and the bolt face.

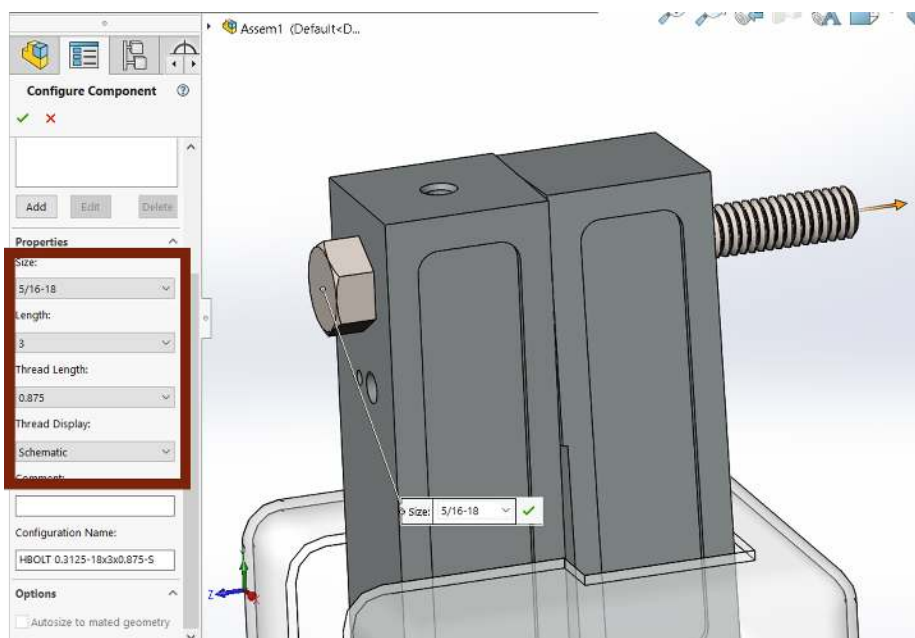


Figure 7.25 Hex Bolt Property Manager

Then navigate to the Plain Washers Type A folder and choose a Preferred Narrow washer to place on the other side of the bolt. Make sure the size matches that of the bolt (5/16) and that you add a concentric and coincident mate locking rotation. Add a second washer, but don't mate it yet.

The difference between Type A Washers and Type B Washers is that Type B Washers are manufactured to a tighter tolerance.

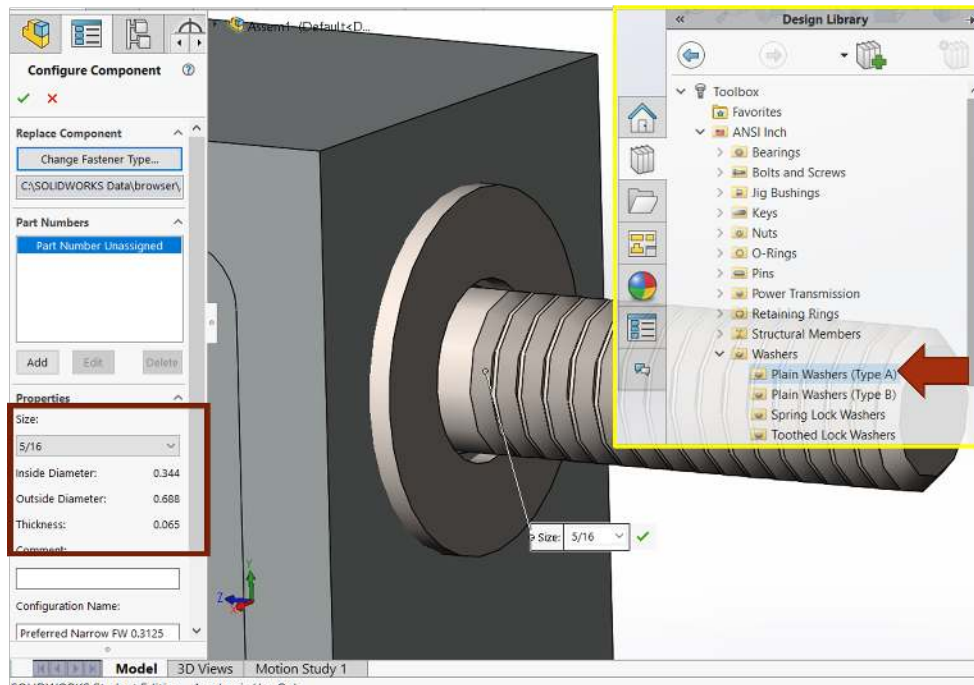


Figure 7.26 Plain Washer Property Manager and Placement

REMAINING SPRING AND FASTENERS

Insert **CAE1008** into the assembly using the Insert Components command. This is the spring that will be flush with the washers and used to apply pressure to the cylinder. This will help prevent locking of the air motor if the cylinder is kept from wiggling during operation.

Apply mates to CAE1008 so that the Top and Right Plane of CAE1008 are coincident with corresponding planes from the hex bolt. Also, apply coincident mates between the end faces of CAE1008 and the corresponding faces on the washers.

Add the two nuts on the end, locking the rotation through the concentric mates.

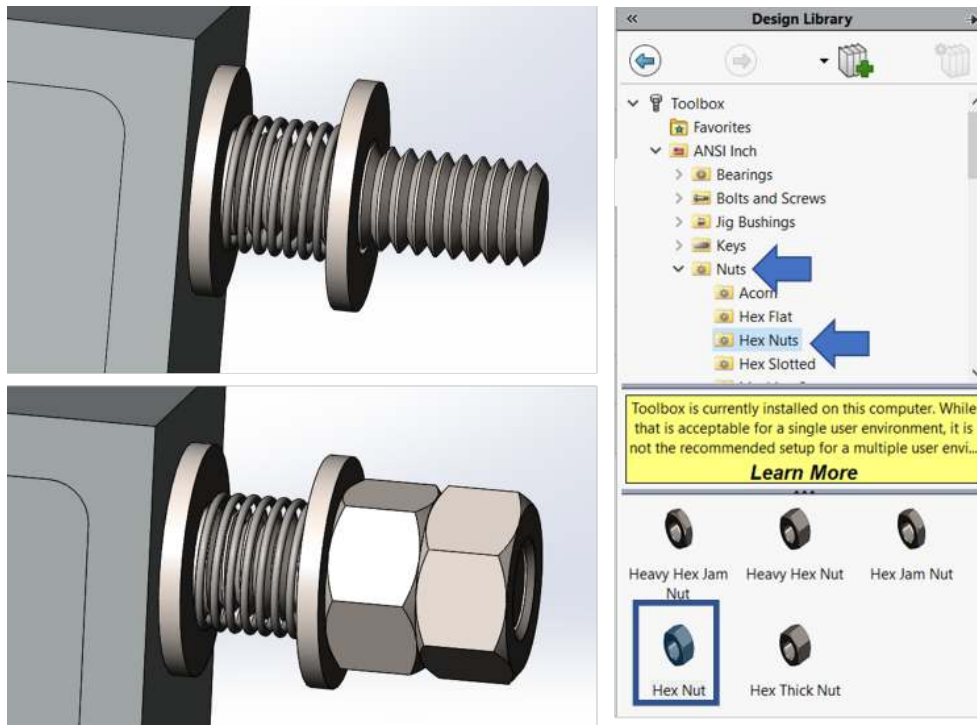


Figure 7.27 Spring and Washer Mates

Add the socket head cap screw on the bottom of the assembly using the design library toolbox. Be sure that there is a concentric mate between the screw and the hole that locks rotation, and a coincident mate between the bottom of the screw head and the bottom of the counterbore.

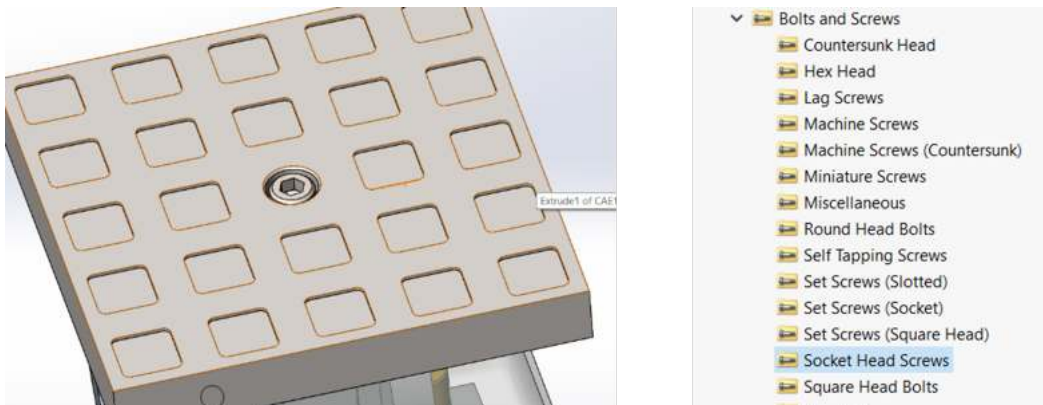


Figure 7.28 Socket Head Screw

The final thing needed to finish the assembly is adding the two-socket head, flat point set screws on the flywheel and the post. Use the design library toolbox to add a #10-32 by .25" length set screw between CAE1004 and CAE1005 and a #10-32 by .375-inch length set screw between CAE1002 and the hex bolt.

To mate the set screws, add a concentric mate between the cone of their bottom and the threaded hole of the mating component as well as a tangent mate between the flat bottom of the set screw and the shaft the set screw will be digging into.

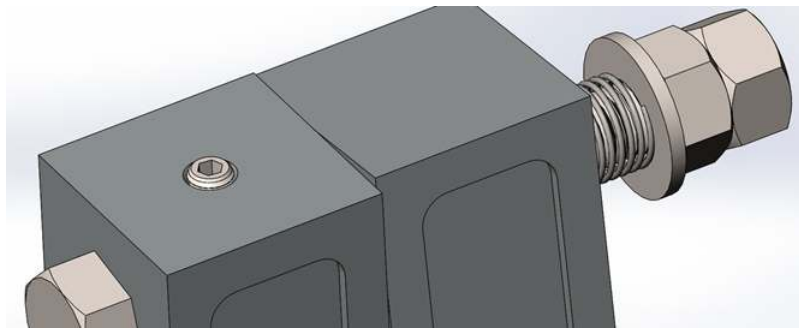


Figure 7.29 Top Set Screw

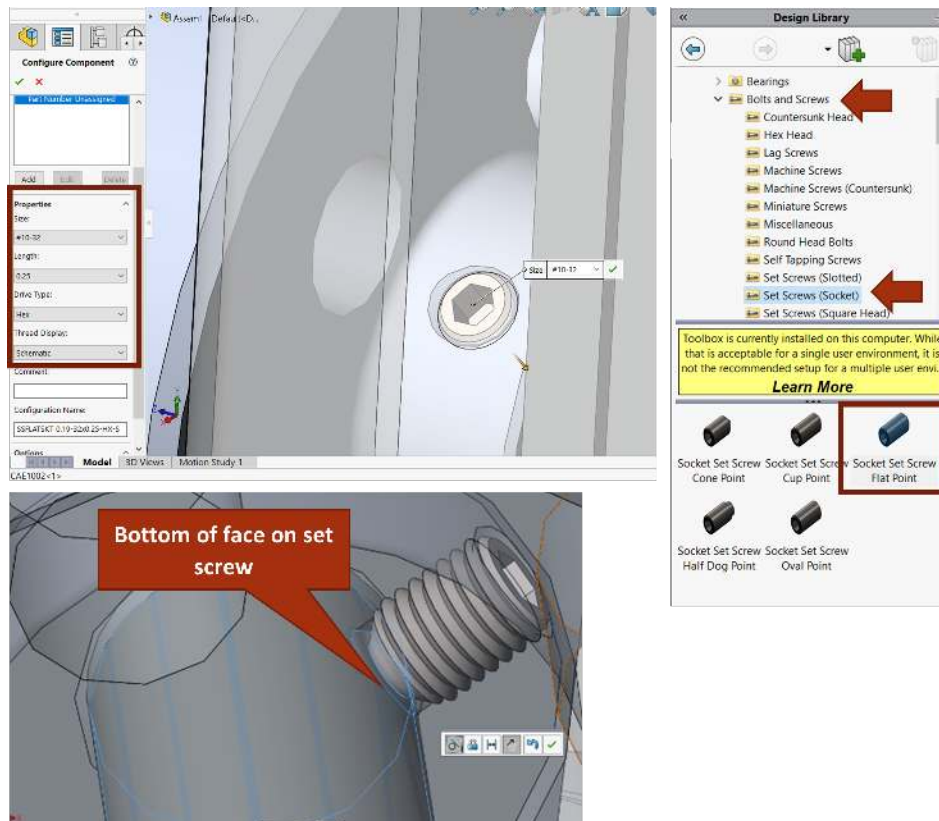


Figure 7.30 Flywheel Set Screw

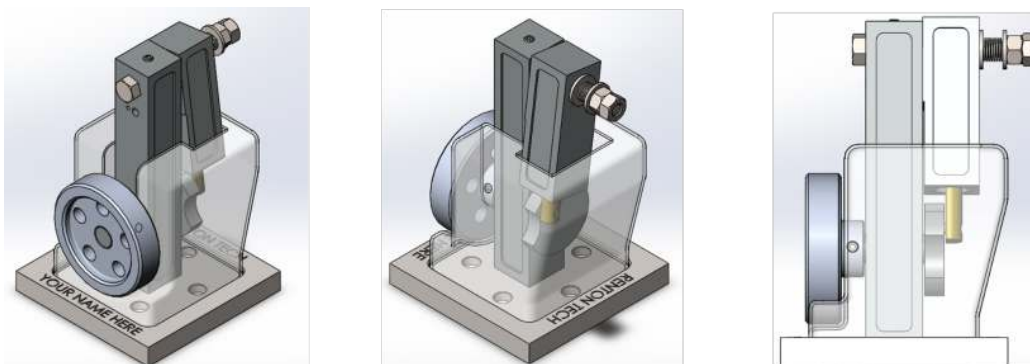


Figure 7.31 Finished CAE1000 Assembly

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7.7: Assembly Evaluation

When creating assemblies in SolidWorks, the purpose is to document how everything goes together and troubleshoot/identify potential problems with the current design allowing for design changes later. We can do this through many tools on the Evaluate Tab of the CommandManager of a SolidWorks Assembly. There are many tools, but we will only cover interference and collision detection here.

INTERFERENCE DETECTION

Interference detection will check the entire assembly model for any components that are interfering with each other. For example, if you have a pin that should have clearance with a hole, but the pin size is too big, you can run interference detection to quickly find this error so that you can correct it.

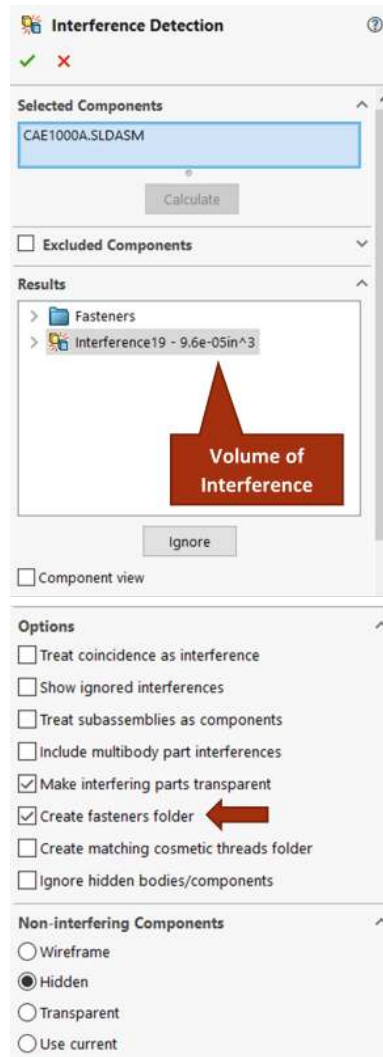


Figure 7.32 Interference Detection Property Manager

To use this tool, you activate it from the Evaluate tab of the CommandManager, set your options in the Property Manager, and then click “Calculate”. If there are interferences in your assembly, they will show up in red as you select them in the property manager list and the volume will show next to it in the property manager. You can analyze interference within the entire assembly or between specific parts. You can also choose to sort fastener interference into its own folder because fasteners will always show as interfering due to the fact that SolidWorks doesn’t model the threads or position fasteners exactly aligned with thread profiles when mated.

When you have many interferences, it can be difficult to identify them so you are able to control how non-interfering components appear in the property manager. In the figure below, all non-interfering components are hidden to really bring out the interfering components. The interference shown however is designed for, representing a press fit. Interference detection is really useful for finding those interferences that aren't designed for.

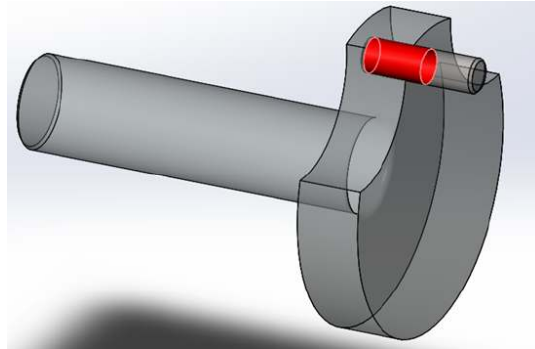


Figure 7.33 Designed Press Fit Interference

COLLISION DETECTION

Collision Detection is another tool you can use specifically for assessing moving parts and how you will assemble the assembly in practice. Collision detection is nested within the Move Component command. To use collision detection, you activate Move Component from the CommandManager, select the Collision Detection option from the Property Manager, assign other desired settings, and then move the components. You can specify only a couple components to be included in the scope of the test or test within the entire assembly.

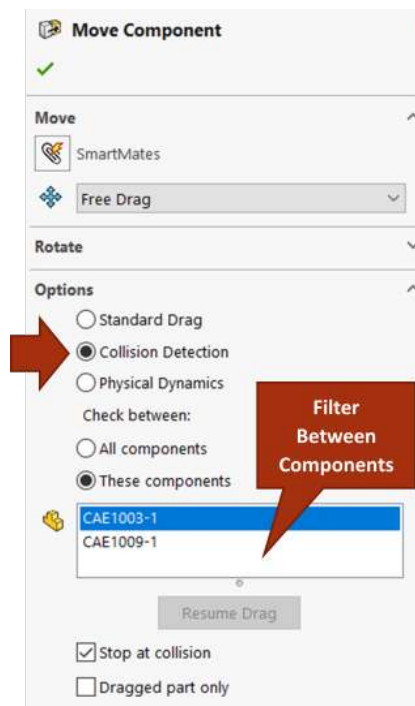


Figure 7.34 Collision Detection in Move Property Manager

Most likely when testing assembly, you will need to suppress mates in the assembly because you've already fixed them. Imagine testing how CAE1009 fits around the rest of the assembly; you need to suppress the mates that are fixing the component in the assembly before you can test it by moving it with collision detection active.

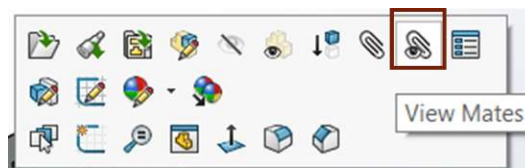


Figure 7.35 View Mates in Context Toolbar

You can suppress mates of a single component by right-clicking the component and choosing “View Mates” (a mate icon with an eye icon) in the context toolbar will show all the mates involving that component in a separate dialog box. You can select the mates you need to suppress from the dialog and suppress them.

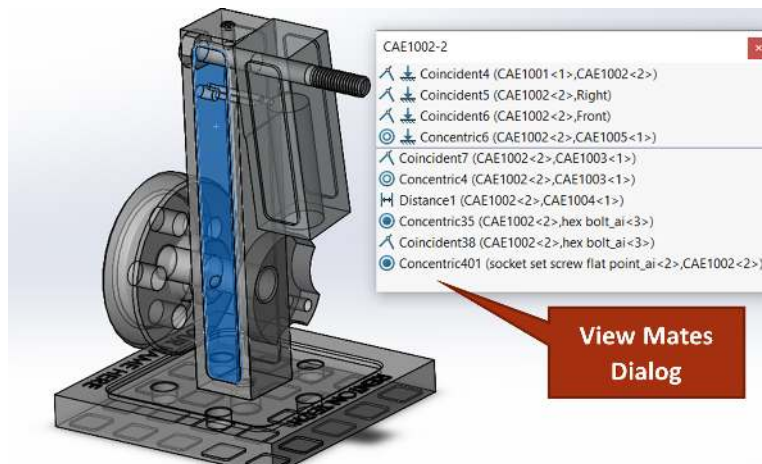


Figure 7.36 View Mates Dialog

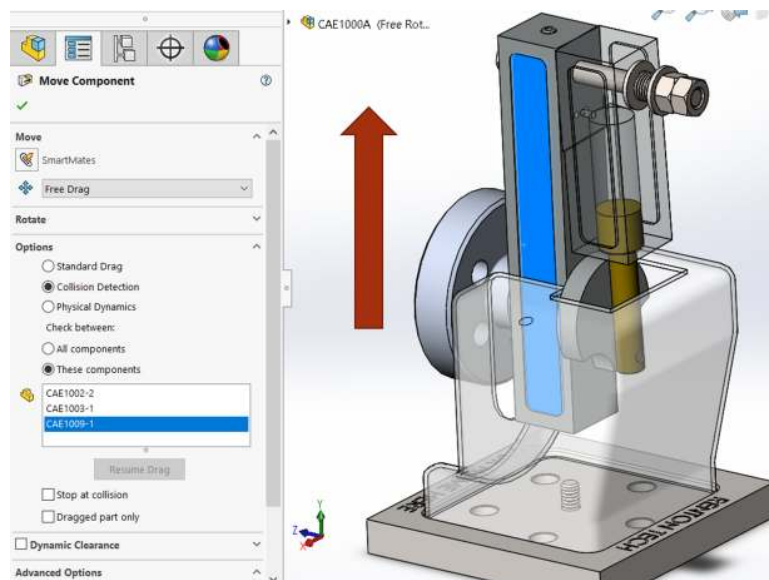


Figure 7.37 Using Collision Detection with CAE1000

? Exercise 7.7 Using Interference and Collision Detection for CAE1000

In this exercise, you will learn how you can use two assembly evaluation tools to determine if your design needs adjustment or if it will function in terms of its design intent. In CAE1000, perform interference detection and evaluate any interferences in the model and also run collision detection to see if the parts can be assembled or if assembly order will be important for this design. You will need to suppress a mate on CAE1002 to run collision detection.



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7.8: Assembly Documentation

Assembly documentation is one of the most important pieces of the product data package because it not only informs assembly personnel on how to assemble the pieces and what pieces are necessary for the product, but it also informs manufacturing of what each part is for, it informs the quality control group on context of how the different parts are to be inspected. Assembly documentation also helps maintenance and repair groups on how the product is assembled when functioning properly and of all the parts involved.

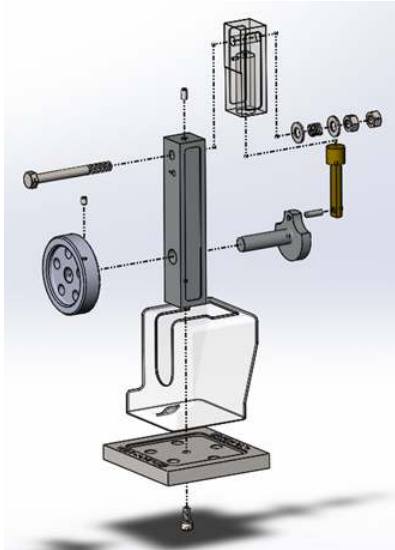


Figure 7.38 Exploded View of CAE1000

The most used piece of assembly documentation is the **Exploded View**, which clearly shows how all the parts go together. You can make notes within the model as well as balloon each component to differentiate the different parts. These annotations are importable to assembly drawings from the assembly.

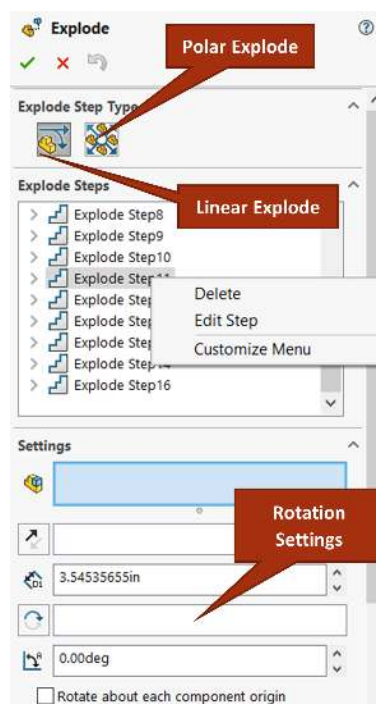


Figure 7.39 Exploded View Property Manager

To make an exploded view, you activate the Exploded View command from the Assembly tab of the CommandManager. From there, you click components you want to move out (you can move more than one component at a time as well) and then use the axial triad to move the component or components. If you wish to rotate the components instead of moving them axially, there are also rotation rings for rotation.

With exploded views, each movement is an exploded step in the exploded view feature. If you need to edit a step, right click the step in the property manager and select “Edit Step”. You follow this same process if you want to delete a step. For parts that are rotated while inserted, such as screws, you can rotate them while they come out from an exploded view.

You can also use Polar Explode which allows you to explode components radially. Think about components in a circular pattern; it would most make sense for those types of components to be exploded radially rather than just linearly.

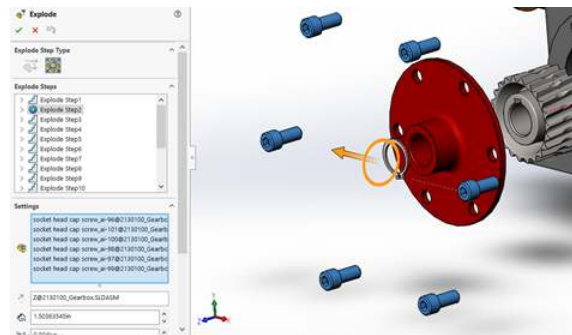


Figure 7.40 Example of Polar Explode

? Exercise 7.8 Creating the Exploded View for CAE1000

In this exercise, you will create an exploded view for CAE1000 using the exploded view property manager. This is going to create a separate configuration of the assembly with all the parts spread out as you specify them. You may also experiment with the exploded line sketch tools to draw phantom lines from part to part to show what connects to what. Also feel free to check out this [Exploded View lesson from MySolidWorks](#).



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CHAPTER OVERVIEW

8: CSWA Preparation

Learning Objectives

The purpose of this chapter is to...

- Identify and apply time-saving techniques to model parts and assemblies faster.
- Practice for the CSWA, getting to the point of being able to complete practice parts/assemblies under 30 minutes.

The CSWA stands for Certified SolidWorks Associate. This is an industry recognized certification that proves your proficiency in using the SolidWorks software just enough to be able to perform mechanical designs. Some would describe this as a certification that proves you have just the right amount of stuff to be dangerous in the mechanical design field.

The [CSWA exam](#) is a 3-hour exam that is taken online which will ask you to model, modify, and assemble parts. To answer questions on the CSWA exam, you will need to know how to access mass property information to identify the correct mass and center of mass of parts and assemblies. The exam is filled with multiple choice and numerical input questions. There are also some drafting competency questions on the CSWA exam.

[8.1: Short Cut Tools](#)

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8.1: Short Cut Tools

Throughout this book, the focus has been on doing commands and selecting entities via mouse selection. However, SolidWorks allows the use of many keyboard shortcuts and mouse gestures to speed up the modeling process. Here we will go over some of the most pivotal techniques which speed up modeling.

Note: All example parts and assemblies you see from this point will be from past CSWA practice exams.

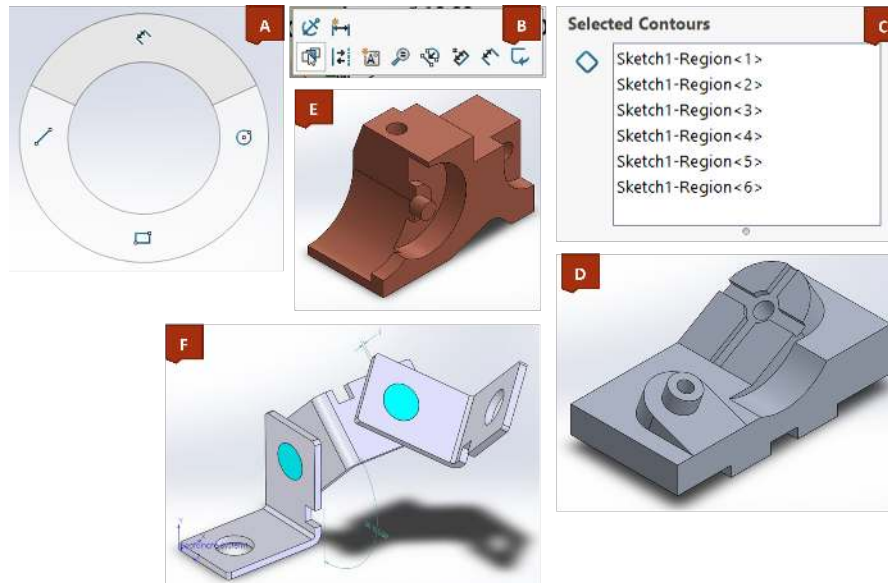


Figure 8.1: Summary of SolidWorks Shortcut Tools. (a) Mouse Gestures; (b) Context Toolbar; (c) Selected Contours Selection Box; (d) Past CSWA Part; (e) Past CSWA Part; (f) Past CSWA Assembly

CONTEXT TOOLBARS

One of the most powerful efficiency tools in SolidWorks is the context toolbar which appears and is customized based off what is selected. You can select multiple entities and use the context toolbar to add relations within a sketch, add mates within an assembly, hide, make transparent, make construction geometry, anything you can think of that appears in the context toolbars already.

To select multiple entities, hold down CTRL on your keyboard and left click the things you wish to select. If you want to unselect something, simply click the selected thing while still holding down control.

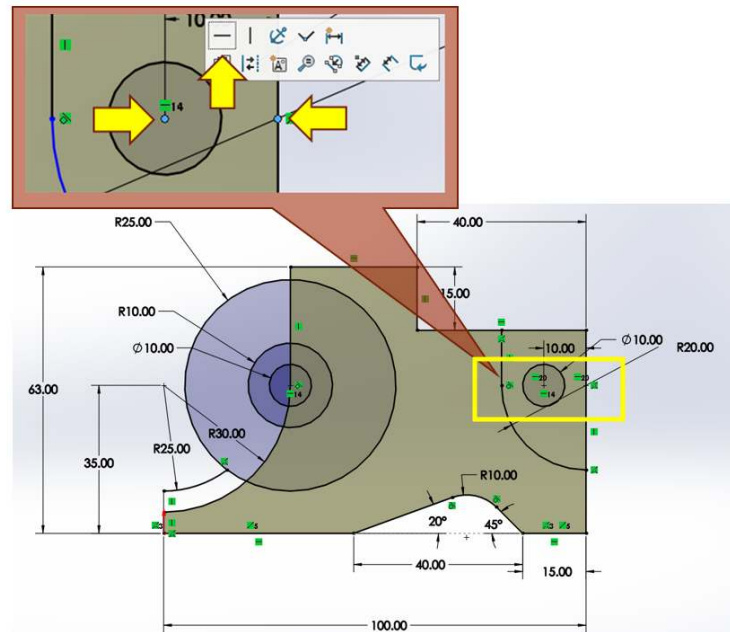


Figure 8.2 CTRL Click Sketch Entities to Quickly Add Relations

MOUSE GESTURES

Mouse gestures is an efficiency tool, like the context toolbars, that gives you access to commonly used commands quicker than accessing them from the CommandManager or the Menu Bars. The mouse gesture wheel is how you use mouse gestures and you can customize the wheel anyway you want. The default mouse gesture wheel was shown at the beginning of the chapter, having the line, circle, rectangle, and dimension tool within the wheel. You can customize any command into the mouse gesture wheel and have as many as 16 mouse gesture commands within the wheel at once.

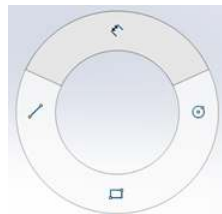


Figure 8.3 Mouse Gesture Wheel

To access the mouse gesture wheel, hold down the Right mouse button and move your mouse in the direction of the gesture you want. As soon as you let go of the right mouse button, the command selected is activated. This reduces the amount of mouse travel needed to activate these common commands in a sketch. When outside the sketch environment, you can change your view orientation in this manner. To customize your mouse gestures for sketches, parts, assemblies, and drawings, go to the options drop down and click “Customize...”. Then go to the Mouse Gestures Tab.

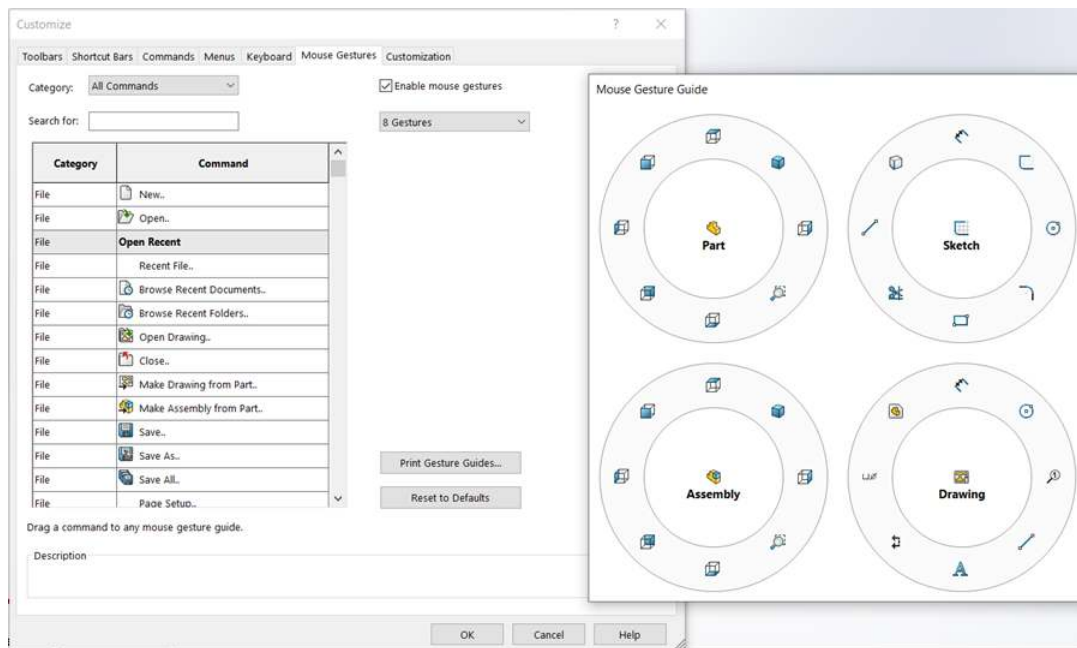


Figure 8.4 Customizing Mouse Gestures

KEYBOARD HOTKEYS

Another way to minimize mouse travel is to use your keyboard to activate commands or open windows close to your mouse cursor. SolidWorks has tons of **keyboard hotkeys** that can be seen from SolidWorks Help browser. Some of the most common keyboard hot keys are CNTRL + numbers 1-8 which will change your view orientation in parts and assemblies. Pressing the spacebar will also bring up the view orientation cube. Another is “S” which brings up a shortcut toolbar and CNTRL + B which rebuilds the current document.



Figure 8.5 Shortcut Menu for CTRL + S

SELECTED CONTOURS

Selected contours are a feature customization tool that gives huge power to your sketches. This tool helps minimize design rework by allowing you to create multiple features from the same sketch. This eliminates the amount of trimming and Convert Entities you need to use when designing complex parts. Whenever you create a feature for a sketch, such as an Extruded Boss/Base, in the Property Manager, there is a selection box for Selected contours. Here you can choose which contours the feature will be applied to. For example, in the sample part shown below, one sketch was used for multiple extruded bosses and extruded cuts by selecting contours per feature.

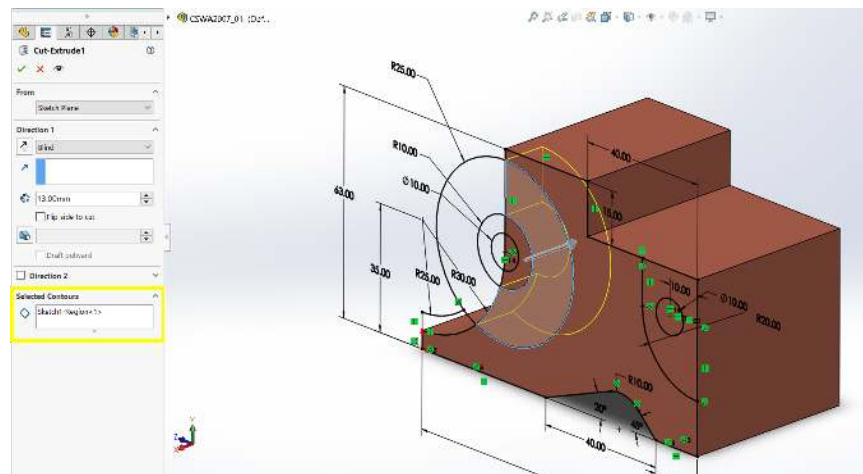


Figure 8.6 Using Selected Contours to Create Multiple Features from the Same Sketch

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Glossary

Add Relation | See [SolidWorks Help: Add Relation](#).

Angle mates | Angle mates are like distance mates except you are specifying an angle between two planar faces, planes, or edges. See [SolidWorks Help: Angle Mates](#).

CAD | Computer Aided design software. SolidWorks, AutoCAD, and SKetchUp are examples of CAD software.

Circular sketch pattern | More information at [SolidWorks Help: Circular Sketch Pattern](#).

coincident | The coincident mate refers to bring two planar surfaces flush or aligned with each other or a surface, edge, or vertex in contact with each other. See [SolidWorks Help: Coincident Mate](#).

Collision Detection | Collision Detection helps you make sure that parts in an assembly don't run into each other or to see when components with collide. See [SolidWorks Help: Collision Detection](#).

Concentric | Concentric mates allow you to align the axis of a curved or cylindrical face with the axis of another. This mate is ideal for aligning holes and other curved faces. See [SolidWorks Help: Concentric Mates](#).

configuration | different versions of the part/assembly.

construction geometry | Also referred to as reference geometry is used to aid the definition of geometry that will become part of the final part, it is a part of the design parameter/constraint that will drive what you see in the final part. See [SolidWorks Help Construction Geometry](#) for more information.

Convert Entities | Convert Entities is a sketch tool that allows you to convert existing model edges visible through a sketch plane into sketch entities such as lines, arcs, and points. See [SolidWorks Help: Convert Entities](#).

design intent | Design intent refers to the geometric constraints that the mechanical designer intends for the model to have. For example, a designer can intend for a hole to be in the center of a plate and constrain it as such in the model. See [More on Design Intent](#).

Design Library | The design library contains many default standard parts that can be used in mechanical design in SolidWorks. You can also add your own standard parts to the design library. See [SolidWorks Help: Design Library](#).

DFM | DFM stands for Design for Manufacturability. There are many feature based tools in SolidWorks that aid the mechanical designer in designing components and assemblies that can be manufactured by common manufacturing processes. Read this [Introduction to DFM article](#) for more information.

display state | different versions of how the part/assembly appears

display style | The display style is a setting that impacts how the model appears in the graphics area (shaded, wireframe, hidden lines visible, etc.)

Distance mates | Distance mates allow you to specify a set distance between model faces, edges, or vertices. See [SolidWorks Help: Distance Mates](#).

eccentric | The axes/centerlines of elements are not aligned.

Exploded View | An exploded view is a view of an assembly with all the parts spread out so that you can see them individually oriented in context of how they fit into the assembly. See [SolidWorks Help: Exploded View](#).

Extend Entities | This does the opposite of Trim Entities, extending geometric entities to a boundary. See [SolidWorks Help: Extend Entities](#).

Extruded Boss/Base command | Creates a 3D feature from a 2D sketch by adding a specified thickness to the sketch in one direction. See [SolidWorks Help: Extruded Boss/Base](#).

Extruded Cut | See more information at [SolidWorks Help: Extruded Cut](#)

Features | Features in SolidWorks are what make up a 3D model. These are things like extrudes, holes, fillets, rounds, etc.

flyouts | Icons usually in the shape of sideways triangles that expand or collapse sub-items.

fully defined | A sketch geometry status that indicates there is enough design parameters constraining the sketch geometry to allow only one possible condition. See Sketch Geometry Status'.

geometric dimensioning and tolerancing | Geometric dimensioning and tolerancing, or GD&T, is a system of defining and communicating engineering tolerances in terms of the functional requirements of a product. It allows for clear definition of engineering dimensional and geometric constraints through a symbolic language scheme.

Geometric relations | Constraints based on geometry conditions such as parallelism, perpendicularity, tangency, coincidence, symmetry, etc. See SolidWorks Help: Sketch Relations.

Hole Wizard | Hole wizard makes it fast and easy to model standard holes in a mechanical design. The macro tool consist of a [Type tab](#), where the hole specifications are set, and a [Positions Tab](#), where the position of the hole(s) is specified. See [SolidWorks Help: Hole Wizard](#) for more information.

Interference detection | Checks for part models that are taking up the same space in an assembly. See [SolidWorks Help: Interference Detection](#).

keyboard hotkeys | Keyboard hotkeys are keys on your keyboard that enable actions without using the mouse such as zoom, menu activation, and commands. See [SolidWorks Help: Keyboard Hotkeys](#).

Limit distance and angle mates | Instead of setting a distance or an angle, you can specify a limit that the part is free to move within. See [SolidWorks Help: Limit Distance and Angles](#).

Linear sketch pattern | More information at [SolidWorks Help: Linear Sketch Pattern](#).

Mates | Mates are the SolidWorks term for part constraints. They are how one part will constrain to another, much like the sketch constraints/relations you are used to. See [SolidWorks Help: Mates](#).

Mechanical mates | Mechanical mates are a category of mates that match the interaction and behavior of mechanical components such as gears, cams, hinges, slots, screws, and universal joints. See SolidWorks Help: Mechanical Mates.

neutral format file types | Neutral format file types are file types that are typically exported from native file types (such as a SolidWorks file) that contain the geometric and attribute data but not the detailed design data that was used to create the model allowing them to be safely shared with stakeholders outside your organization.

Offset entities | Offset entities places geometry out of line by a prescribed distance. See [SolidWorks Help: Offset Entities](#).

Parallel and Perpendicular mates | Parallel and perpendicular mates are exactly as they sound: they allow you to set faces or edges parallel or perpendicular to each other. See [SolidWorks Help: Parallel and Perpendicular Mates](#).

parametric design | Parametric design refers to using relationships, constraints and parameters to control a design to better clarify design intent. See [more on parametric design](#).

Pierce Relation | The pierce relation adds a constraint on a sketch point to intersect with a sketch line that is perpendicular to the originating sketch plane. See [SolidWorks Help: Pierce Relation](#).

Power Trim | More information from [SolidWorks Help: Power Trim](#).

product life cycle | The path, or cycle, which the product takes from idea conception to product termination and disposal. This includes design, manufacturing, quality, maintenance, and every other function of the product across its whole life. See [more on product life cycle](#).

profile | A profile refers to the 2D geometric elements that make up a cross section. See [SolidWorks Help: Profiles](#) for potential Sweep Profiles.

reference geometry | Reference geometry is geometry such as planes, points, and axes that is used for construction of other geometric features but don't actually represent geometry of the model. See [SolidWorks Help: Reference Geometry](#).

Revolved features | A feature that is extruded by from the sketch profile around a selected axis of rotation.

section views | Section views are views of your model cut in a specific location and orientation. They are useful for evaluating internal components of your SolidWorks part or assembly.

Sketch | The sketch tab contains sketch commands that are used to make 2D profiles and shapes that are then made into 3D features.

sketch geometry status' | Sketch geometry status' indicate the condition of constraints on a geometric entity in a SolidWorks sketch. There is under-defined to indicate too little design specification, fully-defined to indicate there is enough design specification, and over-defined to indicate too much design specification (conflicting design parameters). See [SolidWorks Help: Sketch Geometry Status](#) for more information.

sketch relations | Sketch relations constrain sketch geometry geometrically as opposed to dimensionally. Some examples of geometric constraints are parallelism, perpendicularity, tangency, and alignment. Some sketch relations are automatic and others must be added manually in SolidWorks. See [SolidWorks Help: Sketch Relations](#) for more information.

SmartMates | An automated macro in SolidWorks that determines the best mate for two part entities that are held close to each other while holding ALT. It helps add mates faster. See [SolidWorks Help: SmartMates](#).

Standard mates | The most basic mates such as coincident, concentric, tangent, distance and angle. These are used on the CSWA prevalently. See [SolidWorks Help: Standard Mates](#).

Sweep | Sweep refers to extruding a 2D profile along a 2D or 3D path. See [SolidWorks Help: Sweep](#).

Tangent mates | Tangent mates brings at least one cylindrical or curved face or edge in line with another face or edge. See [SolidWorks Help: Tangent Mates](#).

thin-feature | Thin Feature allows you to apply a thickness to a profile. In traditional extrusions, the profile is usually an open contour however with sweep, the profile is a closed contour. See [SolidWorks Help: Thin-Feature](#).

Trim entities | "Trim" away excess geometry. See [SolidWorks Help: Trim Entities](#).

under-defined | A sketch geometry status that indicates there are not enough design parameters constraining the sketch geometry to allow only one possible condition. See Sketch Geometry Status'.

Width Mate | Width mate refers to positioning a pair of planar surfaces centered between another pair of planar surfaces. See [SolidWorks Help: Width Mate](#).

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