

TRDS 160: CAD FOR INDUSTRY



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About the Authors

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Preface

Computer Aided Design (CAD) packages have become ubiquitous in modern manufacturing environments. A typical parametric CAD package can support the design of a standalone component or an assembly of components, and its related engineering drawings. A typical parametric CAD package can also support analysis such as thermal, fluids or finite element-based analysis of mechanical properties. CAD packages even allow for simulation of kinematics, additive manufacturing, subtractive manufacturing, and quality control inspection. Therefore, CAD skills are an important tool in a manufacturing technician's toolbox. However, before a designer can effectively deploy a CAD package for these advanced applications, it is important to understand and become proficient in the use of a CAD package to sketch, model, and assemble a prototype of the widget.

This book has been written to help the novice CAD user gain a grasp of designing using a modern parametric CAD software package. The package used in this book is Autodesk Fusion 360. It is a relatively new package that boasts a user-friendly interface and computational load in addition to being freely available for use in an academic setting. This book introduces the novice designer to a) interface of Fusion 360, b) creating sketches using Fusion 360, c) creating three dimensional solid models using Fusion 360, d) creating an assembly of solid models, and e) creating engineering drawing using solid models.

The hope of this activity-based book is to create a self-explanatory resource to get started with using Autodesk Fusion 360. The video tutorials accompanying this text allow for the novice designer to learn this package at any location at their own pace. This is an open education resource allowing CAD instructors to build upon to meet their classroom needs.

This book would not have been possible without my co-author Meredith Tummeti's painstaking efforts to review the initial text, provide valuable feedback, and make it compatible to the Libretxt platform. I also want to thank the administration at my institution, Centralia College, for their encouragement and providing an excellent academic environment conducive to this collaborative endeavor.

Happy Learning!

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March 31, 2023

CHAPTER OVERVIEW

Modules

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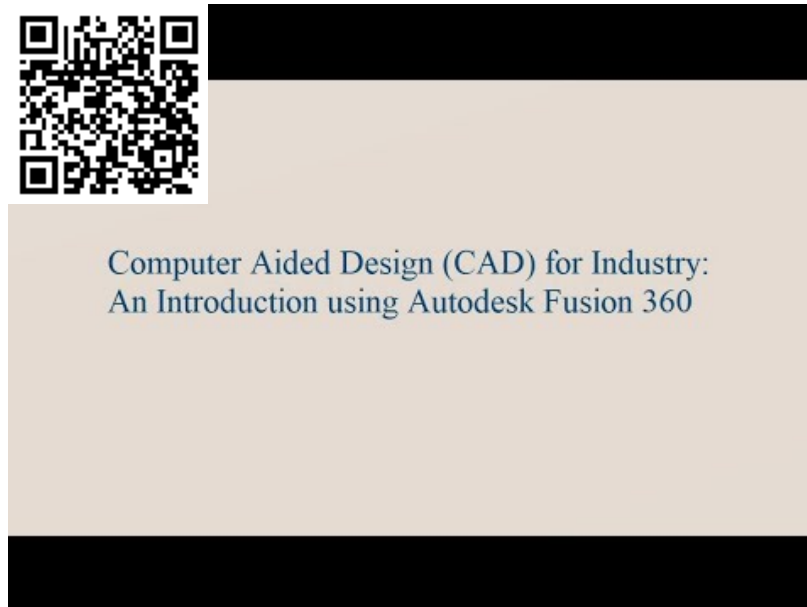
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1.1: Module 1 - Introduction to Computer Aided Design (CAD)

1.1.1 Introduction

Computer Aided Design is the name given to a software package that lets the designer accomplish the following functions: sketch a two-dimensional (2D) sketch, create a three-dimensional body from the two-dimensional sketch, assemble multiple three-dimensional (3D) bodies into an assembly, and create an engineering drawing of the three-dimensional body. These packages typically also come with some analysis packages for structural failure and fluid flow analysis. Out of the multitude of packages available in the market, this book has chosen to focus on Autodesk Fusion 360.

1.1.1.1 Module 1 video:



<https://youtu.be/OulNZgJ94Dw> - Transcript available on YouTube.

1.1.2 Autodesk Fusion 360

Fusion 360 is a 3D CAD package developed by Autodesk especially for students and enthusiasts in the additive manufacturing community. This package is hosted on the cloud. This means that most of the computation and file storage is done on the servers hosted by Autodesk.

1.1.3 System and Connectivity Requirements

In order to use Autodesk Fusion 360 you need to have access to either a personal computer or a laptop that operates on a Microsoft Windows or Apple macOS system. For the Windows, it must run a version of Windows 11 or newer, and for the macOS it must run the version 11 Big Sur or newer. The required internet speed must be at least 2.5 Mbps for download and at least 500 Kbps for upload. More detailed system and connectivity requirements can be found here: [Fusion 360 Help | System requirements for Autodesk Fusion 360 | Autodesk](#).

1.1.4 Installation

You should receive an email from Autodesk with detailed instructions on completing the registration of your account. After that, you will be able to download the Autodesk Fusion 360 application on your desktop. The icon for the application looks like the following image, Figure 1.1.1.



Figure 1.1.1: Autodesk Fusion 360 icon as seen on a desktop.

1.1.5 User Interface

The user interface of the Autodesk Fusion 360 is shown in Figure 1.1.2 below.

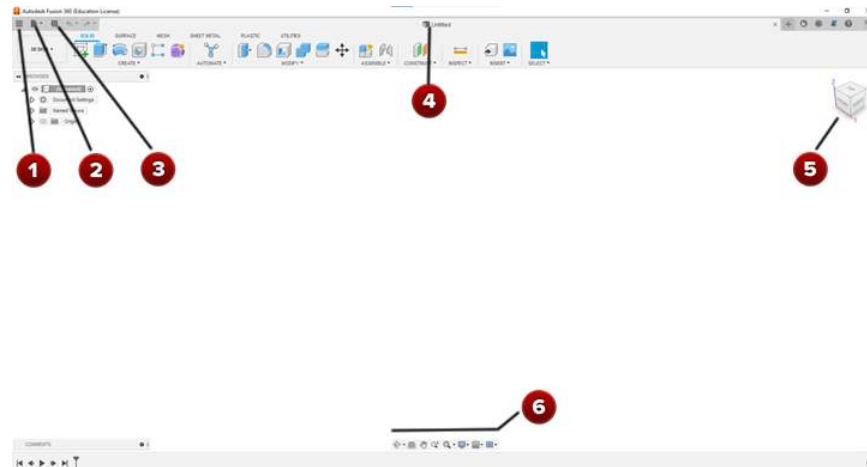


Figure 1.1.2: User Interface for the Autodesk Fusion 360.

1.1.5.1 Bento Menu Button

The Bento menu is labeled number one (1) in Figure 1.1.2 above. By clicking on this menu, the recent files saved on the Fusion 360 cloud can be accessed. A detailed version of the Bento menu is shown in Figure 1.1.3.



Figure 1.1.3: The bento menu button on the Autodesk Fusion 360 interface.

After clicking on the Bento menu button, a sidebar opens up on the left side of the interface. This sidebar contains an icon of the saved file with the saved name and date last modified. Figure 1.1.4 shows this sidebar in more detail.

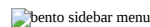


Figure 1.1.4: Sidebar containing saved files.

1.1.5.2 File Drop Down Menu

There is a file drop down menu that is labeled as number two (2) in Figure 1.1.2 above. A detailed version of the file dropdown menu is shown in Figure 1.1.5 and the menu that this button brings up is shown in Figure 1.1.6.



Figure 1.1.5: The file drop down menu on the Autodesk Fusion 360 interface.

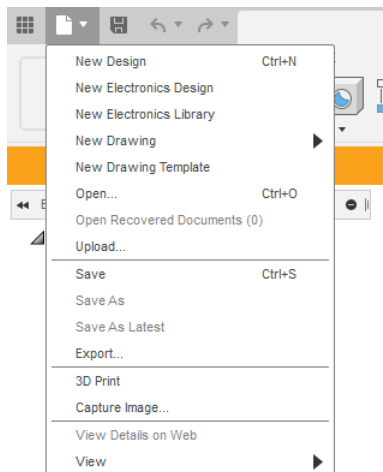


Figure 1.1.6: Options in the file drop down menu.

Within the options in the file drop down menu, the user can create a new design, which refers to a 2D sketch or 3D model. The electronics design will let you create and simulate an electronic circuit that. The new drawing will allow for an engineering drawing to be created. This drawing has to be based on a 3D model that has already been created. The option Open allows for opening of a preexisting file. The Upload option is used to move a file saved on the computer the user's cloud based account with Autodesk. The Save option is used to save a file that the user is currently working on. For a 3D model, the file format is (*.f3d). If the file has never been saved before, it will be saved as a new file and if the file has been saved before, it will let the user overwrite the file. Export allows for a currently open file to be saved in a different file format that the standard Autodesk file format, such as STEP or IGES. This allows the file to be read by software packages other than the one on which the file was created. The 3D Print option prepares the current 3D model to be printed out using a 3D printer. The Capture Image option allows for capturing the current model on the screen and saving it as an image file.

1.1.5.3 Floppy Disk Button

The Floppy Disk shaped button allows the user to save the current project. It is shown in Figure 1.1.7. The floppy disk icon is labeled number three (3) in Figure 1.1.2 above.



Figure 1.1.7: The Floppy Disk button on the Fusion 360 interface.

1.1.5.4 File Name

The name of the file is displayed at the top of the tab and is labeled as number four (4) in Figure 1.1.2.

1.1.5.5 Coordinate System

The Coordinate System is labeled as number five (5) in Figure 1.1.2. This coordinate system is used to indicate the plane in which the user is viewing the projection.

1.1.6 Interacting with the CAD environment

Within the CAD environment, there are ways to manipulate the current model in 3D space. The toolbar labeled number six (6) in Figure (\PageIndex{2}\) is located at the bottom of the interface and allows the user to manipulate the models. These are shown in more detail in Figure 1.1.8.



Figure 1.1.8: Toolbar to manipulate the model in 3D space.

1.1.6.1 Orbit

The Orbit button, shown in Figure 1.1.9, allows for two types of manipulation: free and constrained. In order to use either of these options the user has to hold down the shift button, middle button of the mouse and move the mouse around. Free orbit allows the model to be moved anywhere in 3D space and in any direction but the constrained option only allows the model to be moved around a fixed point in space.



Figure 1.1.9: The Orbit option to manipulate the model.

1.1.6.2 View

The View button brings the selected component in an assembly to the front. See Figure 1.1.10.



Figure 1.1.10: The View option.

1.1.6.3 Pan

The Pan button allows the model to be moved around in 3D space without changing the orientation of the model. See Figure 1.1.11.



Figure 1.1.11: The Pan button.

1.1.6.4 Zoom and Fit

The Zoom In/Out and Fit option allows the user to adjust the display proportion of the selected model. See Figure 1.1.12.



Figure 1.1.12: Zoom and Fit options.

1.1.6.5 Display Settings

The Display settings options allow for the model to be viewed in different backgrounds, camera angles, and lighting options. See Figure 1.1.13.

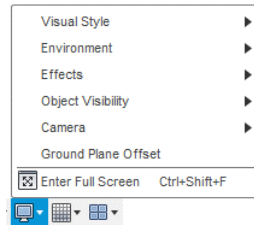


Figure 1.1.13: Display options.

1.1.6.6 Grid and Snaps

The Grids and Snaps options allow for essentially a graph paper functionality while sketching. See Figure 1.1.14.

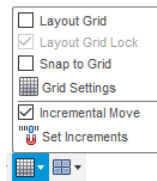


Figure 1.1.14: Grid and Snaps options.

1.1.6.7 Viewports

The Viewports option allows for multiple views to be displayed simultaneously on the current screen. See Figure 1.1.15.



Figure 1.1.15: Viewports option.

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1.2: Module 2 - Sketches

1.2.1 Introduction to Sketches

A two-dimensional (2D) sketch is the basis of any solid model created in a Computer Aided Design (CAD) package. In a new CAD file, click on the create menu and select sketch as Shown in Figure 1.2.1 (left). After clicking on it, the interface for the sketch opens, as shown in Figure 1.2.1 (right). To proceed, a plane must be selected. This plane will become the sketching plane. Note that all three planes are orthogonal to each other.

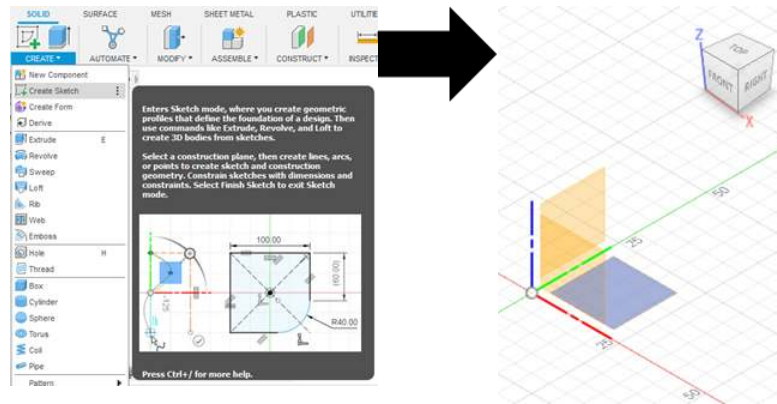
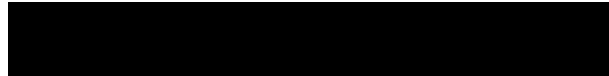


Figure 1.2.1: (Left) Create Sketch option in the Create pull down menu. (Right) Three orthogonal planes to select from to select sketching.

1.2.1.1 Module 2 video, Part 1:



<https://youtu.be/mkAPQLxKB1g> - Transcript available on YouTube.

1.2.2 Sketch Toolbar

Once the sketching plane has been selected, there are multiple methods to sketch on the plane in order to generate the required shape. These tools are shown in Figure 1.2.2 and explained in the following sections.

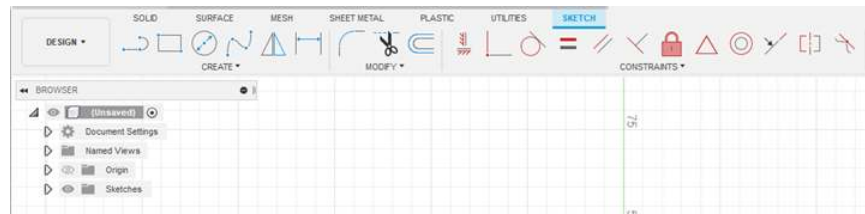


Figure 1.2.2: The Sketch toolbar.

1.2.2.1 Create and Modify Tools

The primary function of the Create and Modify tools is to allow the users to create 2D primitives and edit them.

1.2.2.1.1 Line and Arc

The Line and Arc tool lets the user create a line by default. To create an arc at the end of a line, click the endpoint and drag it to create the arc. (See Figure 1.2.3.)



Figure 1.2.3: Line and Arc icon on the toolbar.

1.2.2.1.2 2-Point Rectangle

This 2-Point Rectangle (Figure 1.2.4) tool lets the user create a rectangle or a square using two points for the diagonal corners.



Figure 1.2.4: 2-Point Rectangle icon on the toolbar.

1.2.2.1.3 Center Diameter Circle

The Center Diameter Circle tool (Figure 1.2.5) lets the user create a circle by defining the center point and the diameter.



Figure 1.2.5: Center Diameter Circle icon on the toolbar.

1.2.2.1.4 Fit Point Spline

The Fit Point Spline tool (Figure 1.2.6) lets the user create a spline through the selected points. A spline is a continuous smooth arc with each consecutive arc being at a tangent to the previous one.



Figure 1.2.6: Fit Point Spline icon on the toolbar.

1.2.2.1.5 Fillet

The Fillet tool (Figure 1.2.7) places an arc of a specified radius at the intersection of two lines or arcs.



Figure 1.2.7: Fillet icon on the toolbar.

1.2.2.1.6 Chamfer

The Chamfer tool (Figure 1.2.8) places an angled edge at the intersection of two lines. It can be defined by a combination of distance and angles.

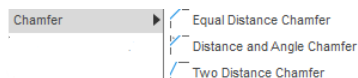


Figure 1.2.8: Chamfer options in the Create and Modify drop down menu.

1.2.2.1.7 Trim

The Trim tool (Figure 1.2.9) trims a sketch curve to the nearest intersecting curve or boundary.



Figure 1.2.9: Trim icon on the toolbar.

1.2.2.1.8 Offset

The Offset tool (Figure 1.2.10) copies the selected curves a specified distance from the original curves.



Figure 1.2.10: Offset icon on the toolbar.

1.2.2.2 Constraints

Once the sketch is partially or fully complete, the elements of the sketch must be constraints so the designer intent can be preserved. Once the sketch is constrained its color scheme turns from blue to black.

1.2.2.2.1 Horizontal and Vertical Constraints

The Horizontal and Vertical Constraints tool (Figure 1.2.11) constrains a single line, or two points, to lie on either the vertical or the horizontal axis, whichever is closer to the current alignment.



Figure 1.2.11: Horizontal and Vertical Controls icon on the toolbar.

1.2.2.2.2 Coincident Constraints

The Coincident Constraints tool (Figure 1.2.12) constrains the position of two points together. It can also work for constraining the position of a point and a line or a curve together.



Figure 1.2.12: Coincident icon on the toolbar.

1.2.2.2.3 Tangent Constraints

The Tangent Constraints tool (Figure 1.2.13) constrains a curve and another object so that they touch at a single point but never cross each other.



Figure 1.2.13: Tangent icon on the toolbar.

1.2.2.2.4 Equal Constraints

The Equal Constraints tool (Figure 1.2.14) constrains similar objects so that their sizes are identical. A change in size in one object will automatically adjust the size of the other component to reflect this change.



Figure 1.2.14: Equal icon on the toolbar.

1.2.2.2.5 Parallel Constraints

The Parallel Constraints tool (Figure 1.2.15) constrains two lines so that they extend in the same direction and do not intersect.



Figure 1.2.15: Parallel icon on the toolbar.

1.2.2.2.6 Perpendicular Constraints

The Perpendicular Constraints tool (Figure 1.2.16) constrains two objects so that they lie perpendicular to each other.

The Fix and Unfix Constraints tool (Figure 1.2.17) constrains the size and location of the point or object. A word of caution, this constrain should be used only if none of the other constrains are pertinent.



The Concentric tool (Figure 1.2.18) constrains two or more arcs, circles, or ellipses to the same center point.



The Colinear tool (Figure 1.2.19) constrains two or more objects so that they share a common line.



The Symmetry tool (Figure 1.2.20) constrains two or more objects so that they are symmetrical.



1.2.3 Objective: Create the following sketch

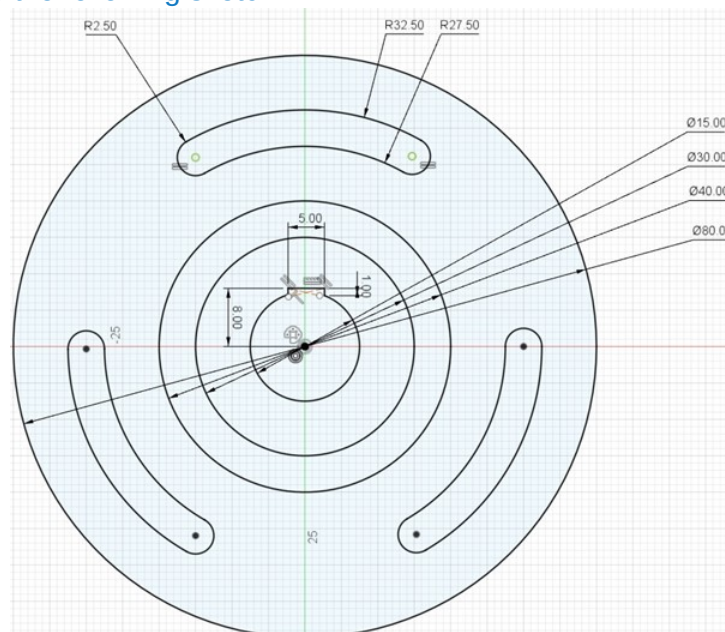


Figure 1.2.21: Final sketch.

1.2.3.1 Module 2 Video, parts 2 & 3



Computer Aided Design for Industry

Module 2 : Sketches

<https://youtu.be/V2N3CMdKeIM> - Transcript available on YouTube.



Computer Aided Design for Industry

Module 2 : Sketches

<https://youtu.be/DuCChzNvzzM> - Transcript available on YouTube.

1.2.3.2 Steps to create the final sketch:

1.2.3.2.1 Step 1 - Create new design

Create a new design and sketch on the front plane.

1.2.3.2.2 Step 2 - Sketch concentric circles

Create our concentric circles with diameter 15 millimeters (mm), 30 mm, 40 mm, and 80 mm. Use the origin as the center point for the circles. Right click on each circle and select sketch dimensions and ensure the dimensions are accurate. Notice that no constraints have been applied to any of the circles and hence their color is green. (See Figure 1.2.22.)

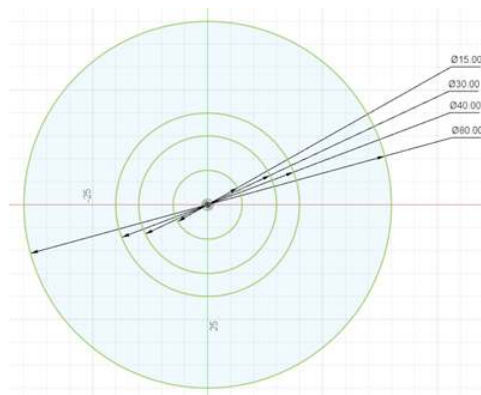


Figure 1.2.22: Step 2.

1.2.3.2.3 Step 3 - Create lines

Next, create a line measuring 8 mm from the center of the circle stretching up along the vertical axis. Next, right click on the line and select construction line. This is done because this line is an aid to creating the sketch and not part of the sketch itself. (See Figure 1.2.23.)

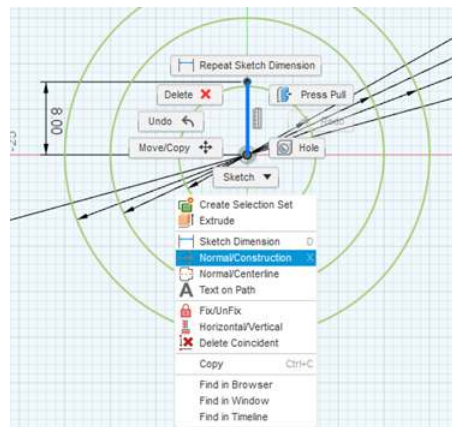


Figure 1.2.23: Step 3.

1.2.3.2.4 Step 4 - Draw lines

At the end point of the line constructed in the previous step, draw a vertical line of length 2.5 mm. Repeat this on the other side. Use the equal to constraint to make sure each line is 2.5 mm and the total length is 5.00 mm. (See Figure 1.2.24.)

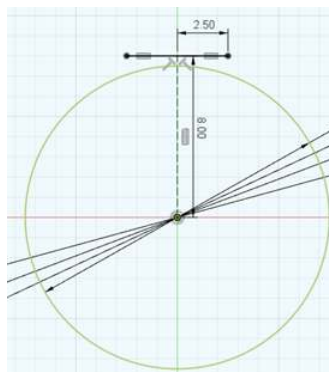


Figure 1.2.24: Step 4.

1.2.3.2.5 Step 5 - Drop and trim line segments

Drop line segments to the circle from the end points of the 5 mm line created in the previous step. Then trim the segment of the circle in between the lines. Also notice that this part of the sketch is now fully constrained, while the rest of the circles are still not constrained. (Figure 1.2.25.)

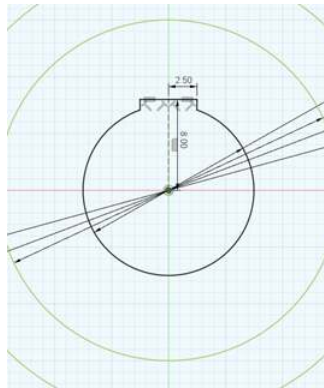


Figure 1.2.25: Step 5.

1.2.3.2.6 Step 6 - Draw lines

Draw two lines at an angle of 60° and 120° from the horizontal as shown in Figure 1.2.26 below. The angle between the two lines is also 60° .

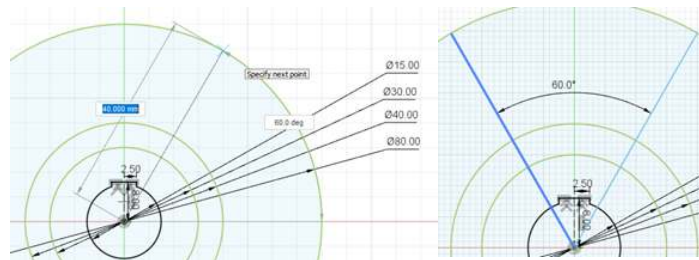


Figure 1.2.26: Step 6.

1.2.3.2.7 Step 7 - Draw concentric circle segments

Draw two concentric circles with diameters 55 mm and 65 mm. Trim the segments of the concentric circles to look like the following figure. (See Figure 1.2.27.)

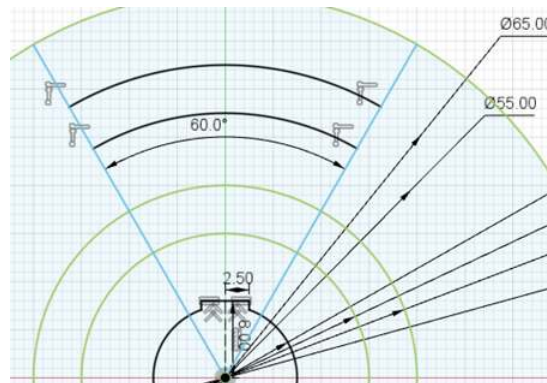


Figure 1.2.27: Step 7.

1.2.3.2.8 Step 8 - Create the three-point arcs

Create a three-point arc on each side of the concentric segments as shown in Figure 1.2.28 below. The radius for each arc is 2.5 mm. You can use equal constraint to ensure the arcs have the same diameter.

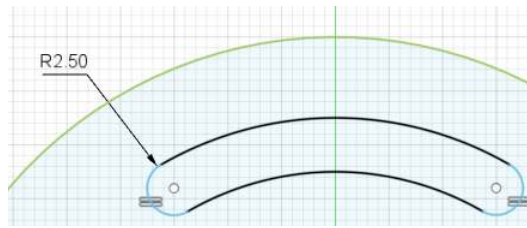


Figure 1.2.28: Step 8.

1.2.3.2.9 Step 9 - Use circular pattern tool

In order to create three such features, use the circular pattern tool that can be found under the create drop down menu. (See Figure 1.2.29.)

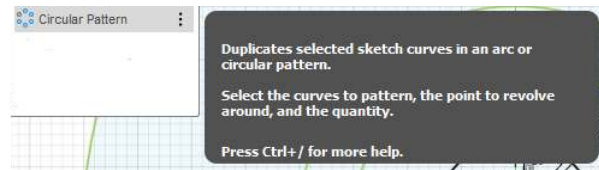


Figure 1.2.29: Step 9.

1.2.3.2.10 Step 10 - Enter parameters

In order to create the pattern, the parameters in the dialogue box are shown in Figure 1.2.30 below. Select all the four (4) arcs that form the slot. The center point is the center of the circle and the total quantity of pattern objects to be created is three (3).

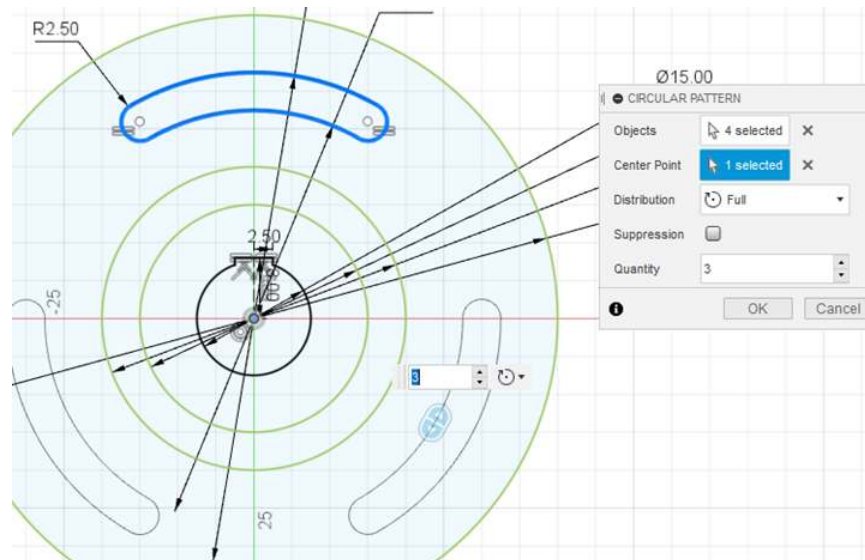


Figure 1.2.30: Step 10.

1.2.3.2.11 Step 11 - Constrain sketch

Use the lock constraint tool to ensure the entire sketch is constrained and the color of all sketch elements should turn black. (See Figure 1.2.31.) Remember to save your work.

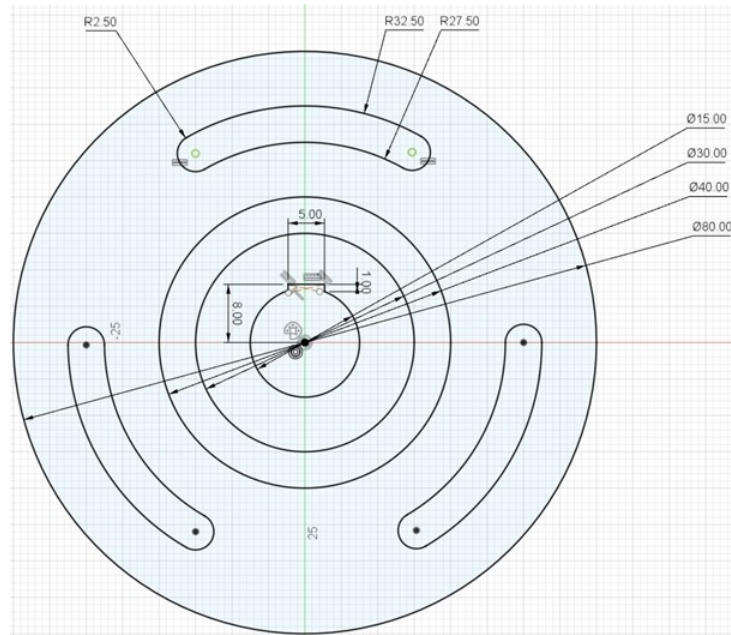


Figure 1.2.31: Final Sketch.

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1.3: Module 3 - Solid Modelling I

1.3.1 Objective

Create a solid model of the sleeve shown in the drawing, Figure 1.3.1, below.

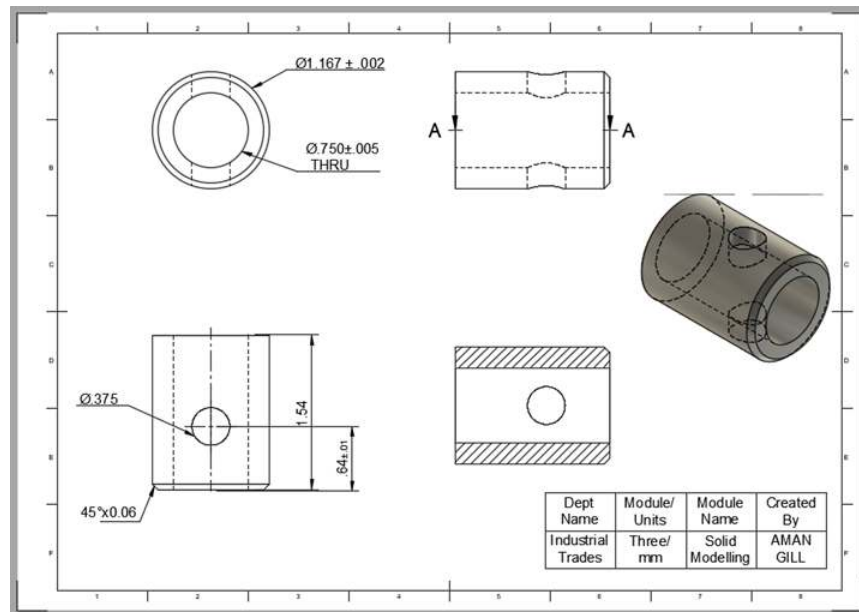


Figure 1.3.1: Engineering drawing of the component that will be produced in Fusion 360 at the end of this module.

1.3.1.1 Introduction

Most objects exist in three-dimensional (3D) space. Modern CAD packages can render 3D objects on 2D screens with the ability to pan and rotate the object to see its 3D projection in real time. The most common commands to produce a 3D object from a 2D object are: Extrude, Revolve, Sweep and Loft. We will go over these tools in the following modules.

1.3.1.2 Module 3 - step-by-step video



Computer Aided Design for Industry

Module 3 : Solid Modelling I

<https://youtu.be/YLdqX2QKZzI> - Transcript available on YouTube.

1.3.2 Create the 3D model shown in Figure 1

1.3.2.1 Step 1: Select the Plane

Open a new File. Make sure the units are millimeters (mm). From the Create dropdown menu, select Create Sketch. Select the Front Plane. Refer to figure 1.3.2 below.



Figure 1.3.2: Step 1 - Selecting the Plane.

1.3.2.2 Step 2: Select the circle shape and create two concentric circles

Refer to the figure 1.3.3 below.

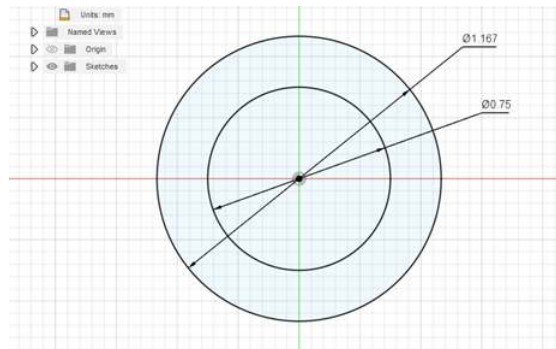


Figure 1.3.3: Step 2 - Selecting circle shape and creating concentric circles.

1.3.2.3 Step 3: Complete the sketch

Click on the green check button labeled finish sketch. See Figure 1.3.4 below.

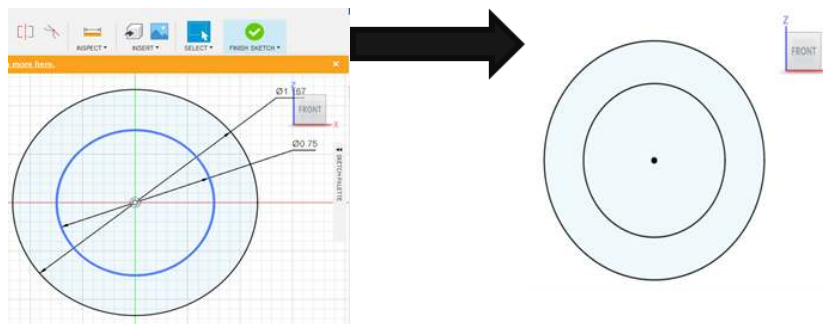


Figure 1.3.4: Step 3 - Complete sketch.

1.3.2.4 Step 4: Extrude the concentric circles

From the create menu, select the extrude option. An edit feature dialogue box, as shown in image will open up. Next, select the outer ring of the sketch just created. The rest of the options in the edit feature dialogue box Direction: One Side; Extent Type: Distance; Distance: 1.54 mm; Taper Angle: 0.0 deg; Operation: New Body. (See Figure 1.3.5 & Figure 1.3.6 below.)

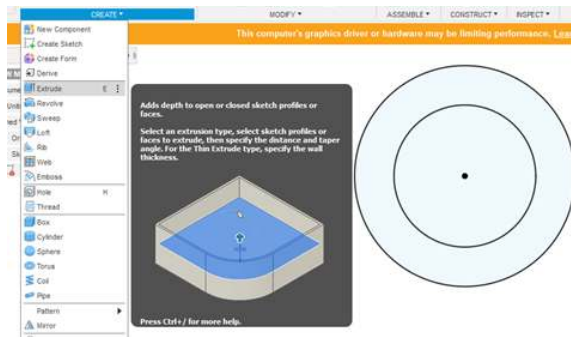


Figure 1.3.5: Step 4 - Extrude option in the Create menu.

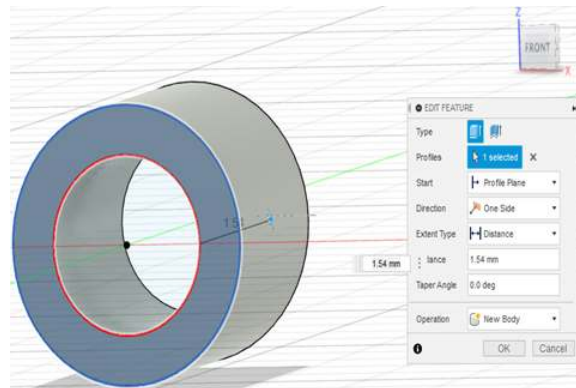


Figure 1.3.6: Parameters for Step 4.

Once the extrude parameters have been entered, select ok.

1.3.2.5 Step 5: Create a chamfer on the outer edge of the cylinder

From the modify menu, select chamfer. Then select the outer edge of the cylinder where you want to place the chamfer. (See Figure 1.3.7.)

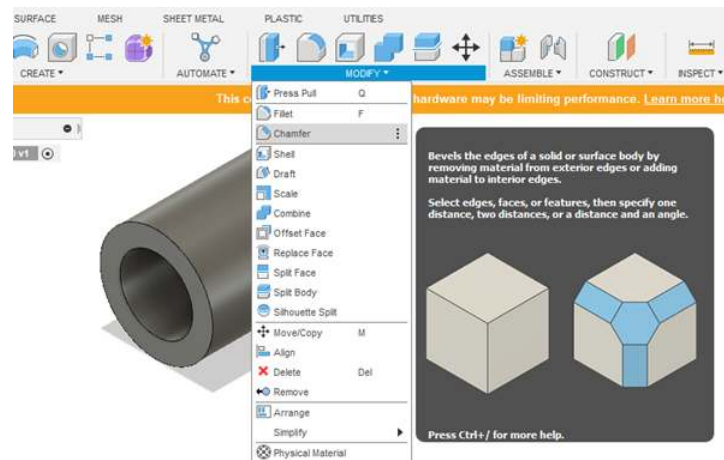


Figure 1.3.7: Chamfer option for Step 5.

The cylinder should now look like the following, Figure 1.3.8.

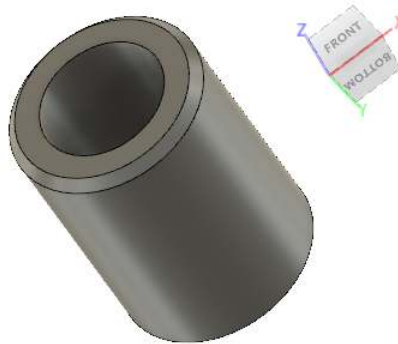


Figure 1.3.8: Completed cylinder with chamfer.

1.3.2.6 Step 6: Create a hole feature in the side of the sleeve

Go to the create menu and select the option labeled “hole”. (See Figure 1.3.9.)

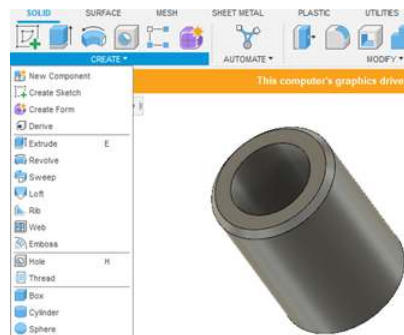


Figure 1.3.9: Step 6 - Create a hole in the side of the sleeve.

Place the hole feature 0.63 mm away from the chamfered lip. The hole feature has a hole type, hole tap type, drill point, and center angle. These options are not critical to create the through hole at hand. However, the Objects to cut must be selected and should be “Body 1”, which in this case the sleeve. (See Figure 1.3.10.)

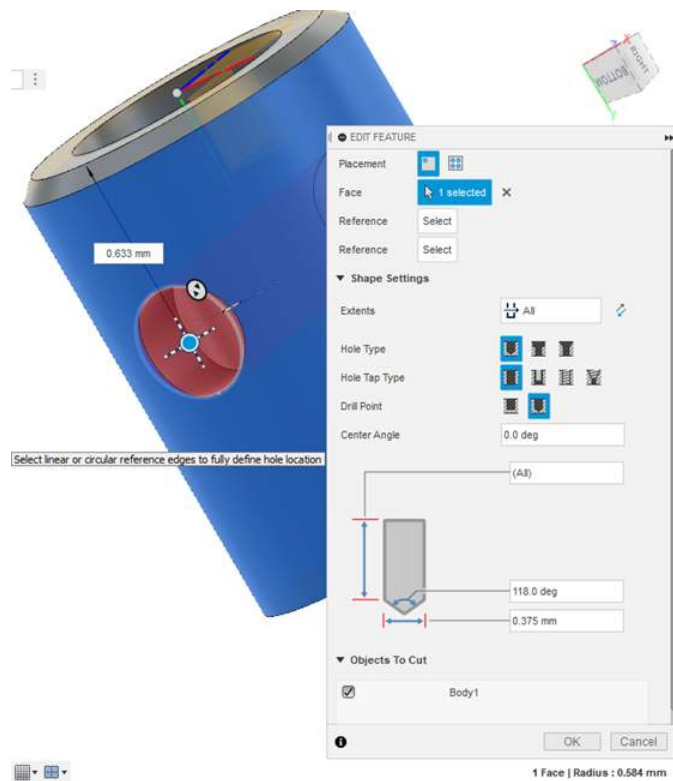


Figure 1.3.10: Choose hole parameters.

1.3.3 Outcome

The final sleeve should look like the following image, Figure 1.3.11. Remember to save your work.



Figure 1.3.11: Outcome of final sleeve.

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1.4: Module 4 - Solid Modelling II

1.4.1 Introduction

This module touches on some more complex features available in the solid modeling toolbox. Let's say you want to add a commercially available fastener component on your drawing. Go to the website of a hardware store and there is usually a drawing or even a CAD model of the fastener. This exercise walks through the steps to generate a CAD model for an M7 hex bolt.

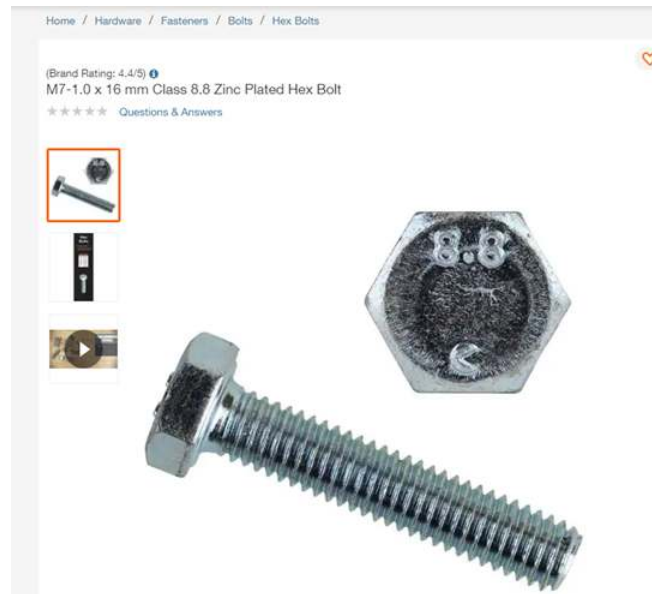


Figure 1.4.1: Screenshot of M7 bolt from the Home Depot website.

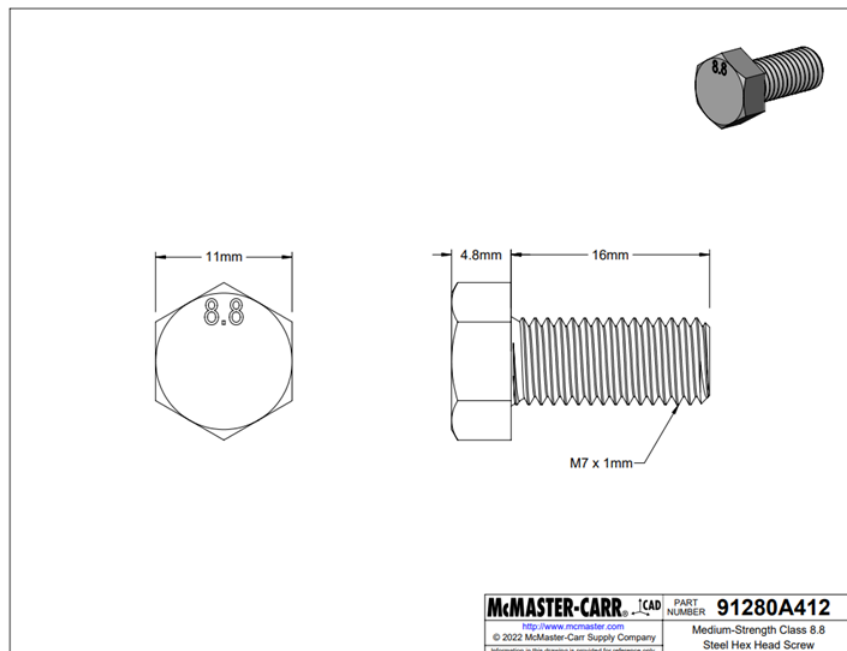


Figure 1.4.2: CAD drawing for an M7 bolt from the McMaster-Carr website.

1.4.1.1 Module 4 - step-by-step video



Computer Aided Design for Industry

Module 4 : Solid Modelling II

<https://youtu.be/oDQKkXrZwxY> - Transcript available on YouTube.

1.4.2 Steps to create an M7 hex bolt model:

1.4.2.1 Step 1 - Create new sketch

Start a new sketch. Make sure the units are in millimeters (mm).

1.4.2.2 Step 2- Draw a circle

On the front plane draw a circle of diameter 7 mm. Then click Finish Sketch. (See Figure 1.4.3.)

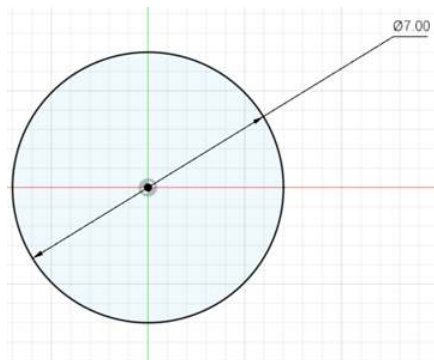


Figure 1.4.3: Step 2 - Draw a circle on the front plane.

1.4.2.3 Step 3 - Extrude circle

From the Create menu select Extrude to create a cylinder using the circle that was just sketched. The extrude length is 16 mm. (See Figure 1.4.4.)

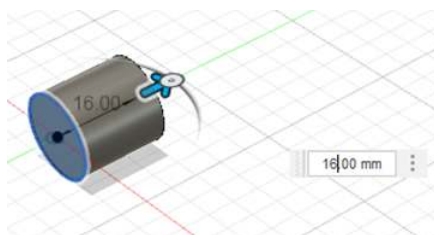


Figure 1.4.4: Step 3 - Extrude the circle into a cylinder.

1.4.2.4 Step 4 - Create threads

From the Create menu, select Threads. Select the surface of the cylinder as shown in the images below. The size of the threads are the same as the diameter and the designation is M7 x 1 (1 mm being the pitch and 7 mm being the diameter). 6g is the tolerance and the direction of the thread is right-handed. This step will produce threads on the stem of the bolt. (See Figures 1.4.5, 1.4.6, and 1.4.7.)

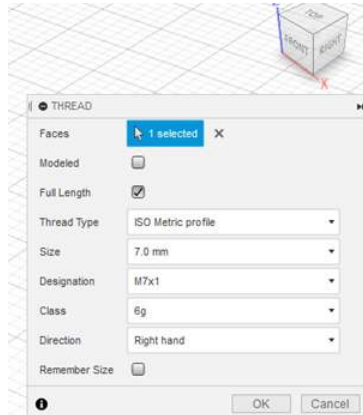


Figure 1.4.5: Step 4 parameters set up for threads.

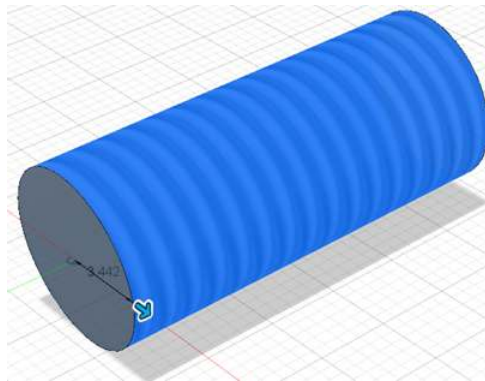


Figure 1.4.6: Step 4 - The cylinder as parameters are being edited.

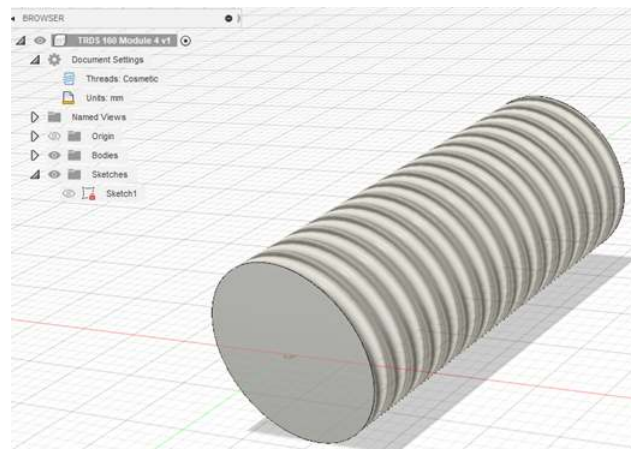


Figure 1.4.7: End result after Step 4 has been completed.

1.4.2.5 Step 5 - Sketch a polygon

Now, to create the head of the hex bolt sketch a hexagon on either face of the cylinder as shown in the image below. Inside the sketch menu, select Polygon > Circumscribed Polygon. There are 6 sides to the polygon (making it a hexagon); the radius of the circle inscribed inside the hexagon is 5.5 mm. (See Figure 1.4.8.)

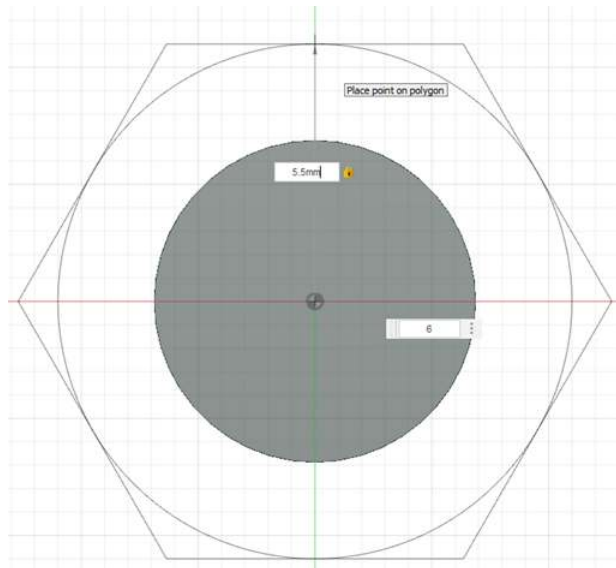


Figure 1.4.8: Step 5 - Sketching the hexagon.

1.4.2.6 Step 6 - Fix the sketch

Best Practice: To fix this Sketch, apply a horizontal constraint to one of the two horizontal sides of the hexagon. This is shown in Figure 1.4.9 below.

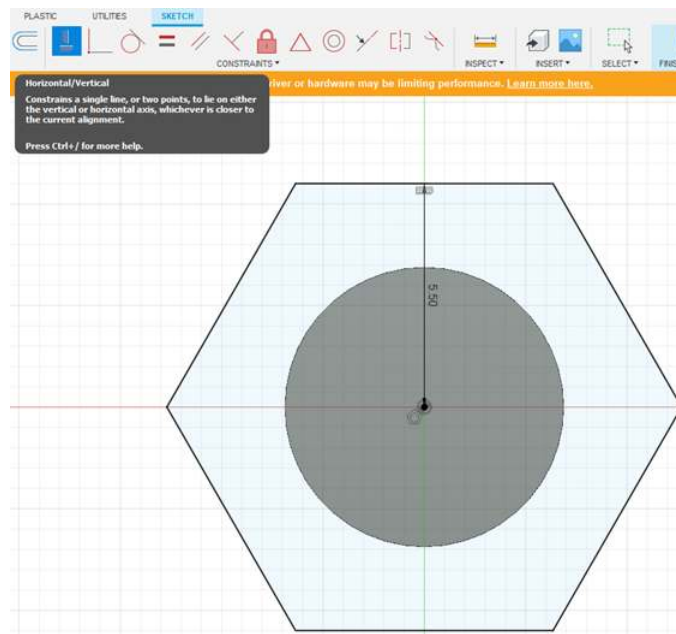


Figure 1.4.9: Step 6 - Apply a horizontal constraint to fix the hexagon.

1.4.2.7 Step 7 - Extrude hexagon

To form the head of the hex bolt, Create > Extrude the hexagon sketch that has been created in the previous step. The extrude length is 4.8 mm as shown in Figure 1.4.10 below. Make sure you select “New Body” in the Extrude menu.

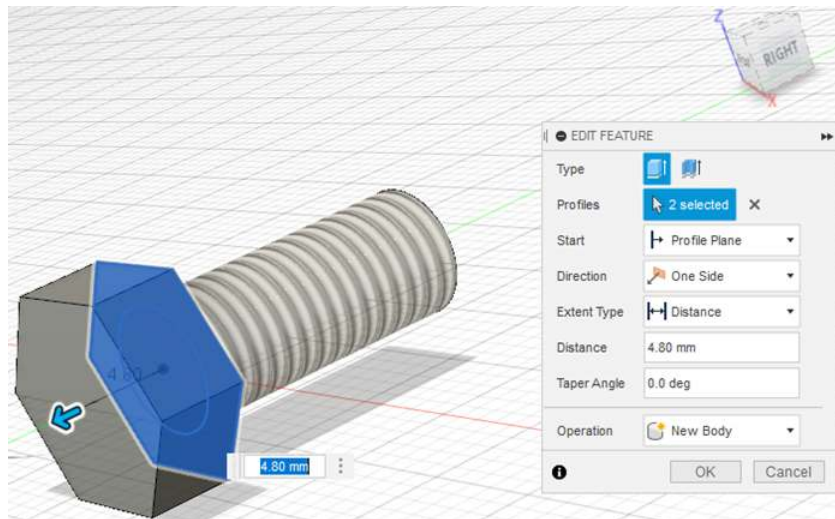


Figure 1.4.10: Step 7 - Extrude hexagon.

1.4.2.8 Step 8 - Sketch a circle on the hex head

In order to create the chamfers on the hex head, do this: sketch a circle of diameter 11 mm on the top face of the hex head as shown in Figure 1.4.11 below.

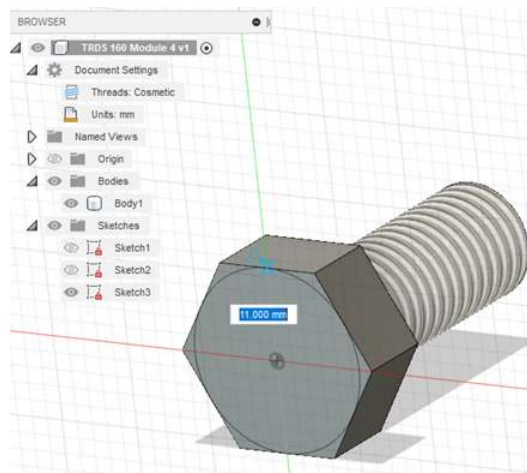


Figure 1.4.11: Step 8 - Sketch a circle on the hex head.

1.4.2.9 Step 9 - Extrude chamfer

After the sketch, got to Create > Extrude and extrude out this sketch as per the parameters shown in dialog box in Figure 1.4.12 below. Extrude out the sketch to a distance of -4.80 mm, and with a taper angle of 45.0 deg. Make sure the type of Operation selected is Intersect.

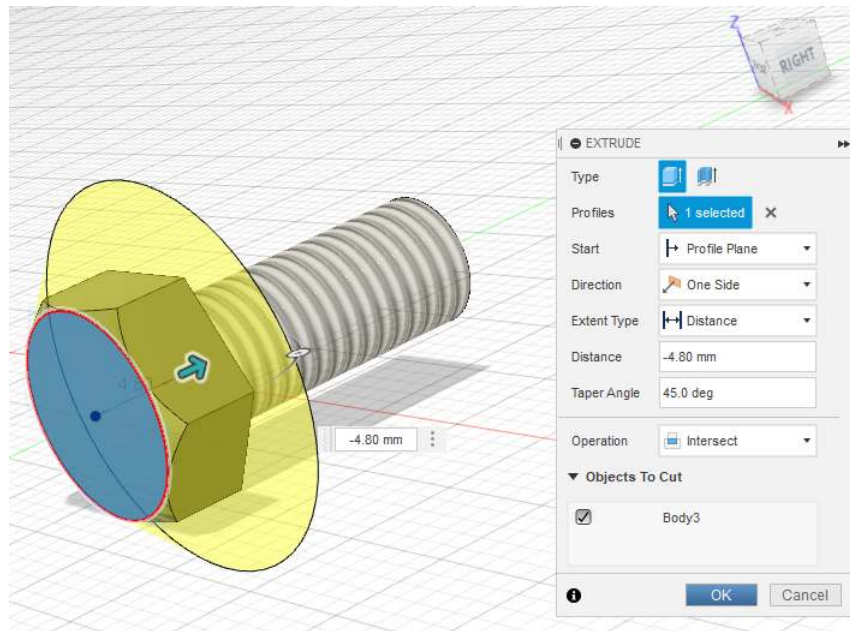


Figure 1.4.12: Step 9 - Creating the chamfer.

At this point, the bolt head should look like the following Figure 1.4.13.

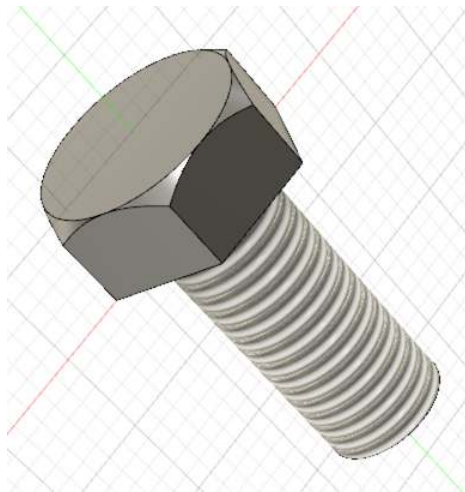


Figure 1.4.13: Result of step 9.

1.4.2.10 Step 10 - Sketch text on hex head

To write “8.8” on the hex head, select the top surface of the hex head and sketch on it. Go to Create > Sketch. Write “8.8” on the head of the hex head. The height of this text box is 1.00 mm and is placed approximately close (eyeball this, no precise location needed) to one of the edge of the hexagon. This is shown in Figure 1.4.14 and Figure 1.4.15 below.

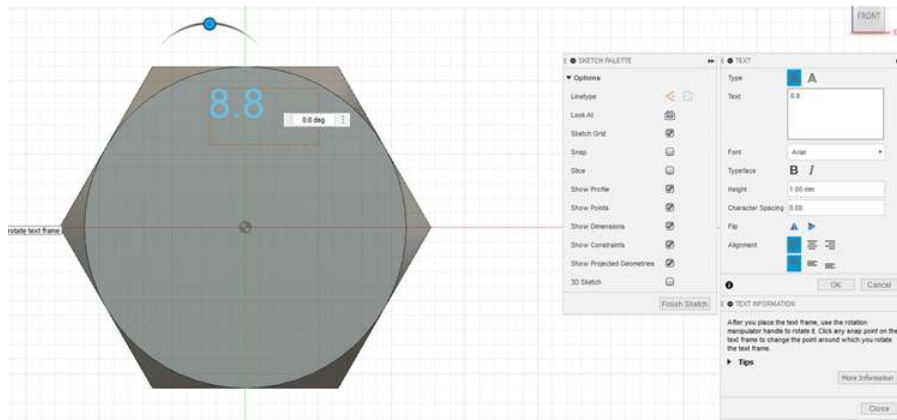


Figure 1.4.14: Step 10 - sketching text on hex head.

1.4.2.11 Step 11 - Emboss text

Now, go to Create > Emboss and select “8.8” to emboss. The height of this feature is 0.10 mm. (See Figure 1.4.15.)

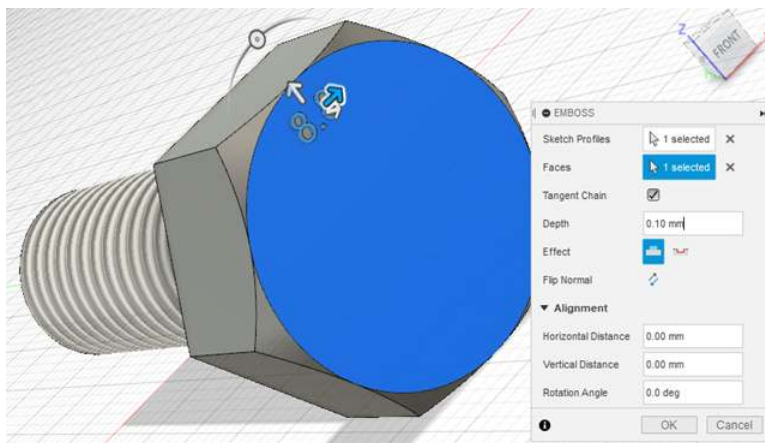


Figure 1.4.15: Step 11 - Emboss text on the hex head.

1.4.3 Outcome

You have successfully created the M7 – 1.0 x 16 mm metric bolt. It should look like the image below (Figure 1.4.16), exactly similar to the Home Depot website, and McMaster Carr website. Remember to save your work.

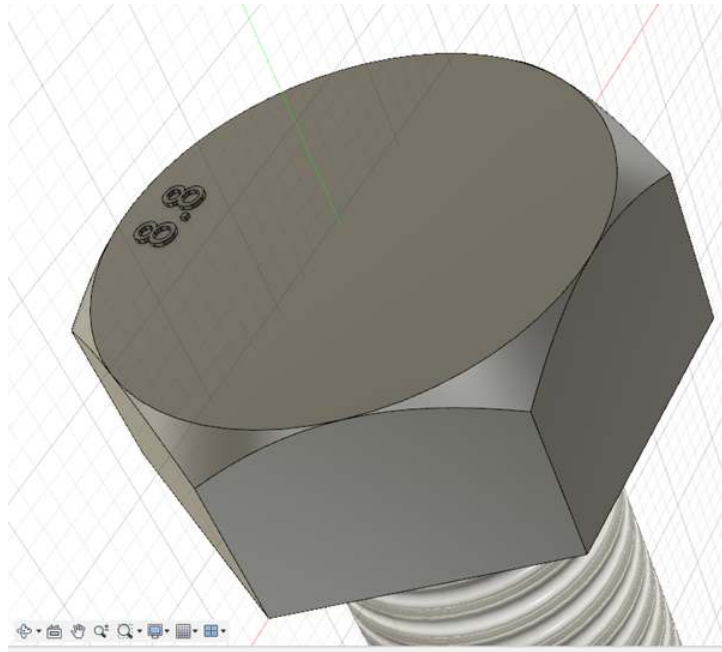


Figure 1.4.16: Result of step 12 and completion of project.

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1.5: Midterm Project - Advanced Solid Modeling

1.5.1 Introduction

An HVAC duct is used to circulate air in a building. This module shows the steps to create an HVAC duct by using advanced solid modeling features in Fusion 360 as shown in Figure 1.5.1.



Figure 1.5.1: HVAC duct that will be created in this module.

1.5.2 Steps to create the HVAC duct

This section lists the steps needed to create an HVAC duct as shown in Figure 1.5.1.

1.5.2.1 Step 1 - Sketch the front face

Draw a “center rectangle” 75 millimeters (mm) in length and 35 mm width on the front plane as shown in Figure 1.5.2. The center of this rectangle is at the origin.

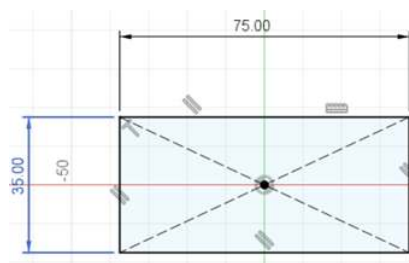


Figure 1.5.2: Initial sketch of the front face.

1.5.2.2 Step 2 - Create an offset plane

Create an offset plane from the bottom plane. This new plane is offset 100 mm away from the bottom plane as shown in Figure 1.5.3.

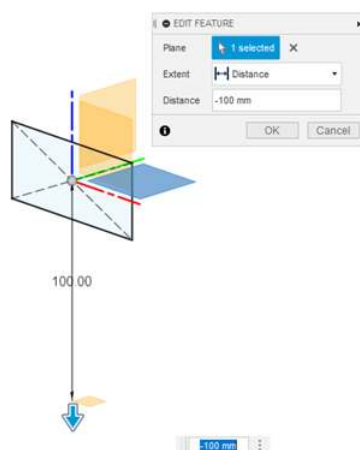


Figure 1.5.3: Creating offset plane from the bottom plane.

1.5.2.3 Step 3 - Create a circle on the offset plane

Create a circle of diameter 50 mm. The center of this circle is 100 mm away from the origin on this new plane as shown in Figure 1.5.4.

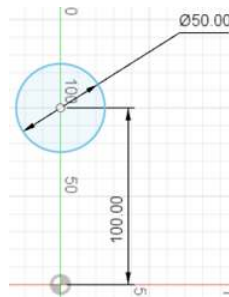


Figure 1.5.4: Circle on the offset plane.

1.5.2.4 Step 4 - Create a parallel plane to the right plane

Create a plane that is parallel to the right plane and coincides with the origin and the center point of the rectangle created in step 1. This plane is in the blue color as is shown in Figure 1.5.5.

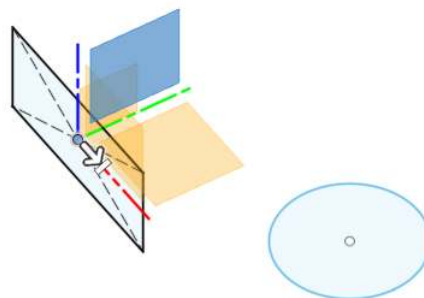


Figure 1.5.5: Parallel plane created in step 4.

1.5.2.5 Step 5 - Sketch a spline

On the plane created in the previous step, sketch a spline as shown in Figure 1.5.6. The choice of the curve is up to the designer, the only restriction is that it must be as smooth as possible and must join the center of the circle to the center of the rectangle.

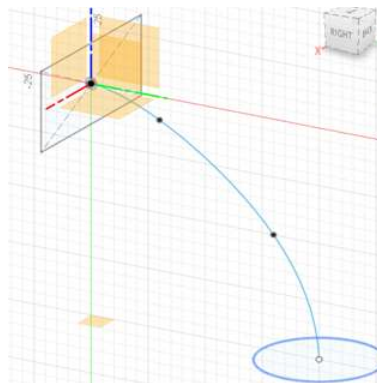


Figure 1.5.6: Sketch the spline.

1.5.2.6 Step 6 - Create the Loft

Under the Create menu, select the “Loft” command and a dialog box will open up. As shown in Figure 1.5.7, under the profiles section, select the rectangle and the circle. To guide the loft profile, select the centerline as the spline you have already created in step 5. Once the profile is created, select OK in the dialog box.

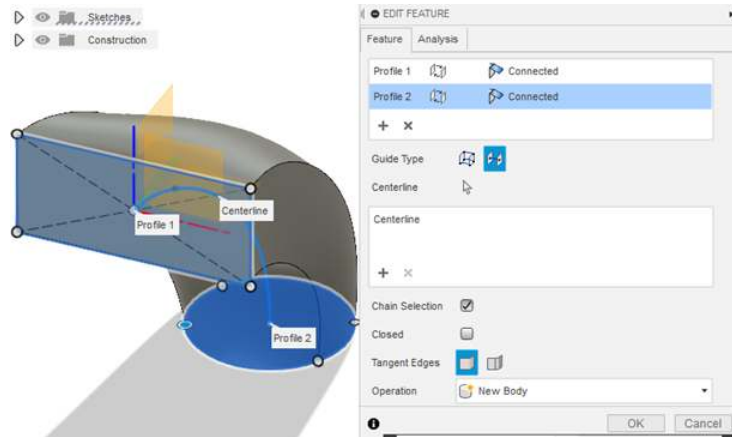


Figure 1.5.7: Create the Loft.

1.5.2.7 Step 7 - Create the Shell

Use the Shell command to hollow out the body created in the last step. Select the two faces (rectangle and circle) and select the thickness as 5 mm as shown in Figure 1.5.8. Remember to save your work.

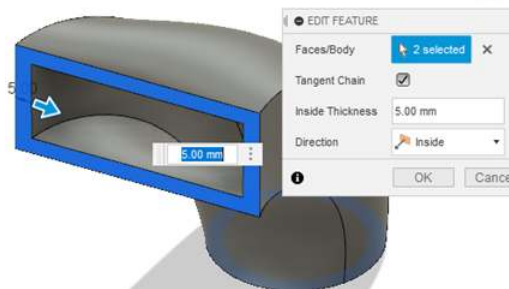


Figure 1.5.8: Create the Shell.

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1.6: Module 5 - Assembly

1.6.1 Introduction

Autodesk Fusion 360 allows for the assembly of multiple 3D components to form a new system of components. This system is referred to as an assembly or subassembly.

As an example, we will assemble a square base plate and a circular plate using five (5) pins as shown in Figure 1.6.1.

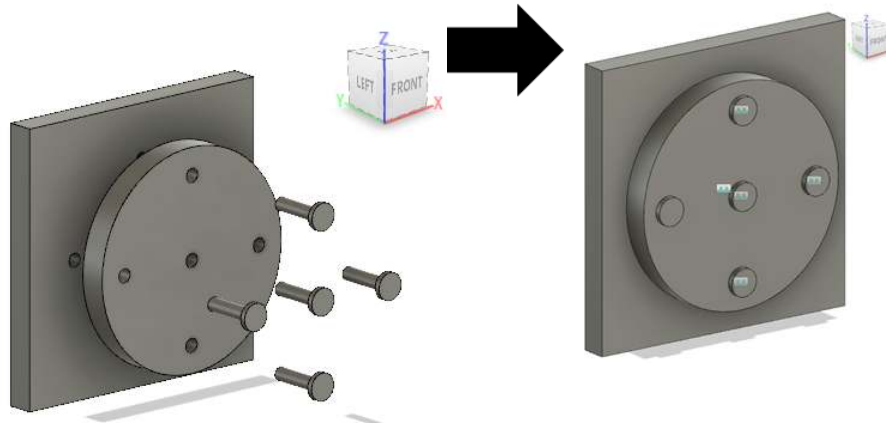


Figure 1.6.1: Objective assembly for this module.

1.6.1.1 Module 5 video showing Assembly options



Computer Aided Design for Industry
Module 5 : Assembly

Part 1 - <https://youtu.be/oMgkWJQgC00> - Transcript available on YouTube.

1.6.1.2 Assembly Options

To assemble various components together, there are a few options available. Some of the more frequently used are discussed in the sections below and are shown in Figure 1.6.2.

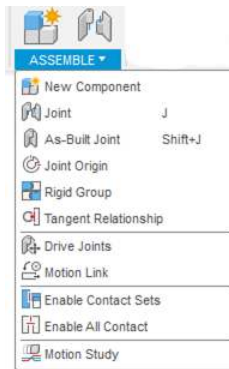


Figure 1.6.2: Assembly options in Fusion 360 - Assemble dropdown menu.

1.6.1.2.1 New Component

The New Component tool allows for adding a component or creating a new component that will be used in the assembly.

1.6.1.2.2 Joint

The Joint tool allows for adding various components relative to each other in the assembly and also specify relative motion between these components. (See Figure 1.6.3.)

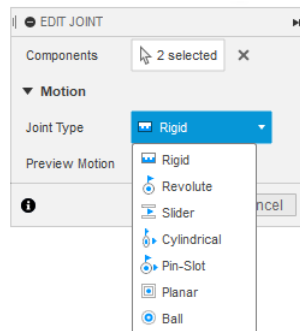


Figure 1.6.3: Sub-options for the assembly option Joint.

Joint Type options for the Joint are as follows:

- **Rigid** locks all components together and removes all degrees of freedom.
- **Revolute** rotates the component around the joint origin.
- **Slides** allows for sliding a component along an axis.
- **Cylindrical** rotates and slides a component on a single axis.
- **Pin Slot** rotates and slides the component on two different axes.
- **Planar** rotates a component around one axis normal to a plane and slides it along two axes parallel to the plane.
- **Ball** rotates a component on all three axes on a gimbal system.

We will only use the “rigid” option today and do a quick demo to understand what other options are available to assemble components. The other options will be used in the following modules.

1.6.1.2.3 As-Built Joint

The As-Built Joint tool positions components next to each other and defines the relative motion. The components maintain their original position.

1.6.1.2.4 Joint Origin

The Joint Origin tool positions the joint origin on the component. Joint origins defines the geometry used to relate a joint's components.

1.6.1.2.5 Rigid Group

The Rigid Group tool locks the relative position of the selected components. Once this is applied, the components act as a single component.

1.6.1.2.6 Drive Joints

The Drive Joints tool specifies the rotation angle or distance value for the joint or degrees of freedom. This option usually works in conjunction with Joint assembly options.

1.6.2 Create and assemble components

The next sections will walk through the steps to create and assemble all three components. The dimensions for these components are also shown in the instructions below.

1.6.2.1 Module 5 step-by-step videos, Parts 2 - 4



Computer Aided Design for Industry
Module 5 : Assembly

Part 2 - <https://youtu.be/1k0z3SDSpP8> - Transcript is available on YouTube.



Computer Aided Design for Industry
Module 5 : Assembly

Part 3 - <https://youtu.be/Hz3an8QqM-A> - Transcript is available on YouTube.



Computer Aided Design for Industry

Module 5 : Assembly

Part 4 - <https://youtu.be/J05dM5DB42I> - Transcript is available on YouTube.

1.6.3 Create the Circular Plate

1.6.3.1 Step 1 - Sketch circular plate

To create the circular plate, create a sketch of a circle of diameter 40 millimeters (mm) on the front plane. (See Figure 1.6.4.)

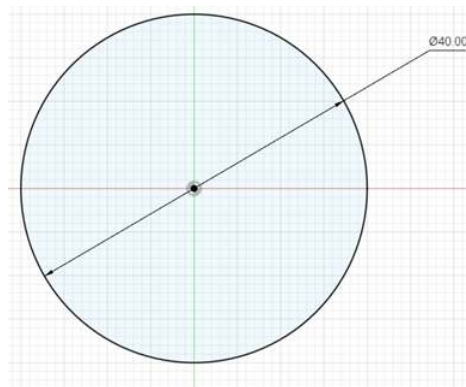


Figure 1.6.4: Circular Plate - Step 1.

1.6.3.2 Step 2 - Extrude circular plate

Extrude the circular plate. The extrude length is 5 mm. (See Figure 1.6.5.)

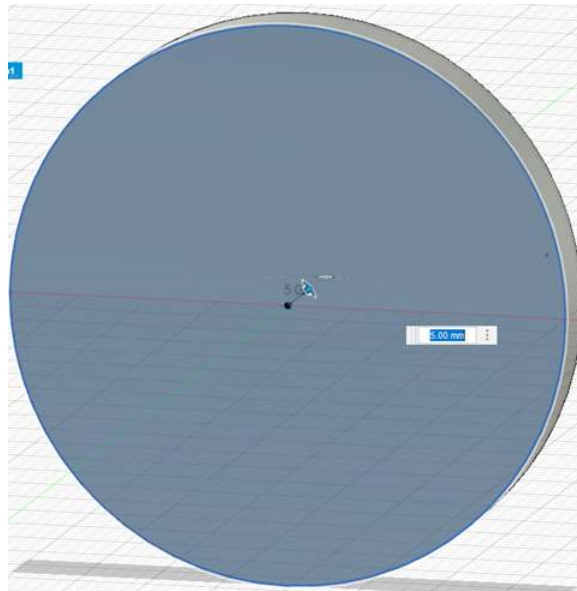


Figure 1.6.5: Circular Plate - Step 2.

1.6.3.3 Step 3 - Sketch 5 circles

Sketch five different circles of diameter 2.5 mm on the face of the circular plate. Sketch the first circle at the center of the circular plate. Then use the pattern command to create four circles at a distance of 15 mm away from the center circle. Then use the circular pattern command to create four circles. The objects selected are the last circle just created. The center point is the center of the circular plate. (See Figure 1.6.6.)

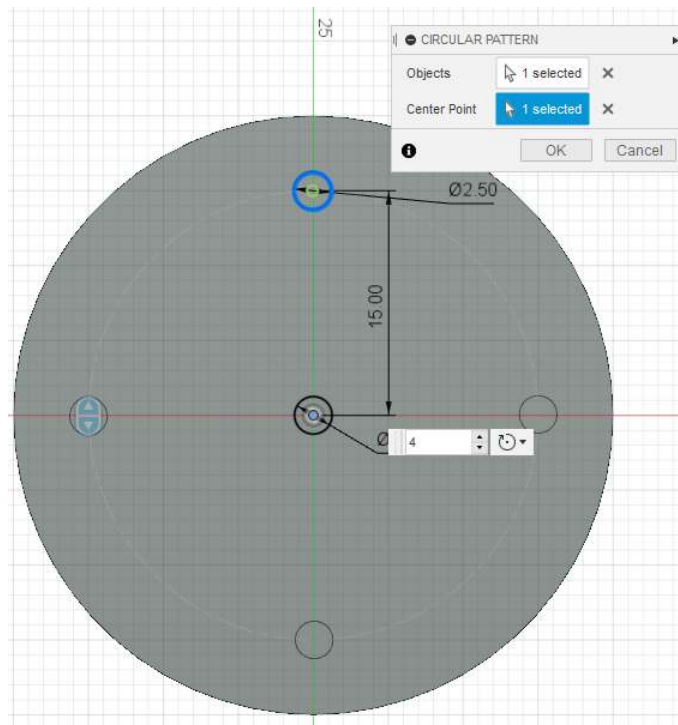


Figure 1.6.6: Circular plate - Step 3.

1.6.3.4 Step 4 - Extrude cut holes

Use extrude-cut to create five holes using the circles sketched in the previous step. (See Figure 1.6.7.)

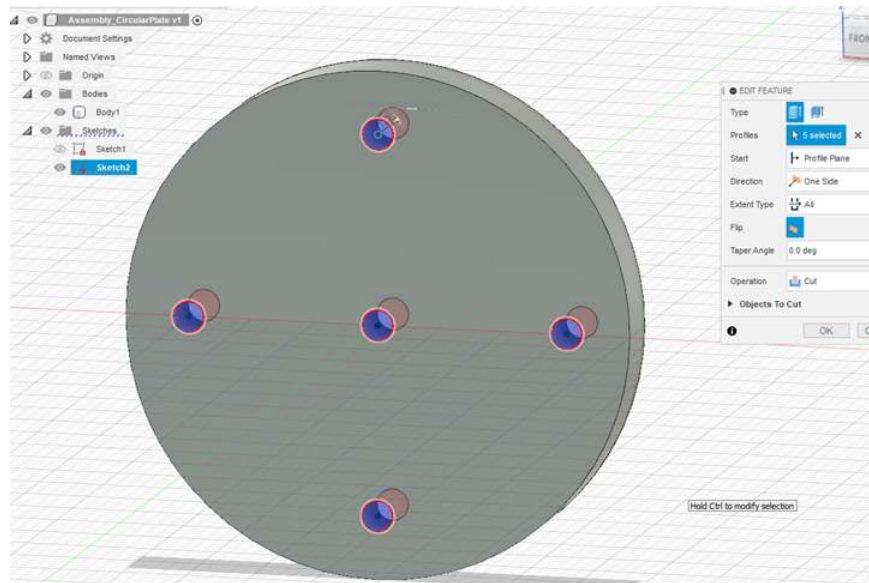


Figure 1.6.7: Circular plate - Step 4.

1.6.3.5 Step 5 - Save circular plate

Save the plate created in the previous step. (See Figure 1.6.8.)



Figure 1.6.8: Circular Plate - Step 5.

Create the Base Square Plate

1.6.3.6 Step 1 - Sketch square base plate

Create a sketch for a base square plate. It is a square with sides of 50 mm each. You can also create a sketch of the five circles that will eventually become holes in the plate. The holes' dimensions and positions are shown in the figure 1.6.9 below and are similar to the ones created on the circular plate, using the pattern command to create four circles at a distance of 15 mm away from the center circle. They are separated equally at an angle of 90°. (See Figure 1.6.9.)

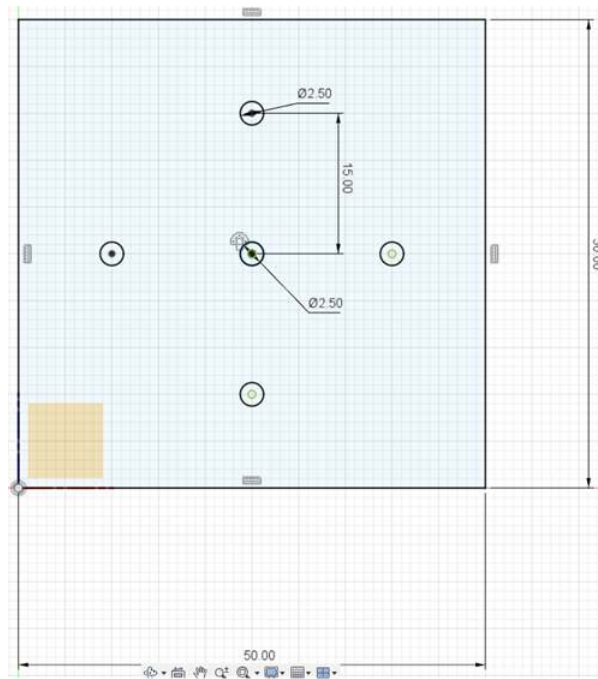


Figure 1.6.9: Base Square Plate - Step 1.

1.6.3.7 Step 2 - Extrude square

Extrude the square created in the previous step. The extrude length is 5 mm. (See Figure 1.6.10.)

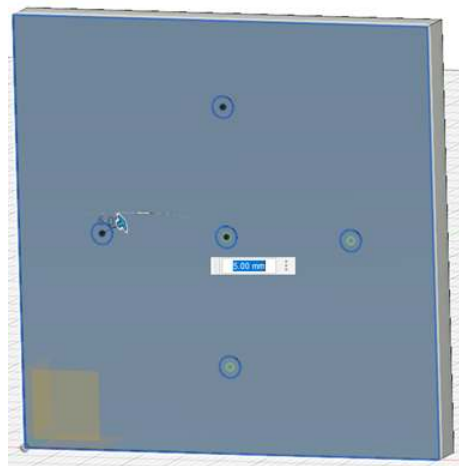


Figure 1.6.10: Base Square Plate - Step 2.

1.6.3.8 Step 3 - Extrude cut holes

Use Extrude Cut to create holes using the sketches of a circle created in the previous steps. (See Figure 1.6.11.)

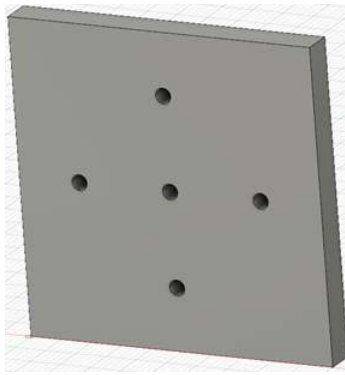


Figure 1.6.11: Base Square Plate - Step 3.

1.6.3.9 Step 4 - Save file

Save the base plate created in the previous steps. (See Figure 1.6.12.)

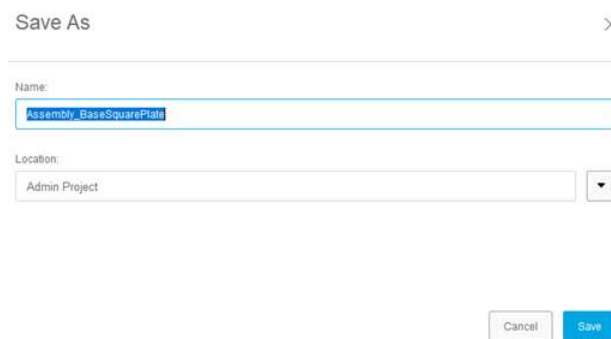


Figure 1.6.12: Base Square Plate - Step 4.

1.6.4 Create Pin

1.6.4.1 Step 1 - Sketch a circle

Sketch a circle of diameter 5 mm on the front plane. (See Figure 1.6.13.)

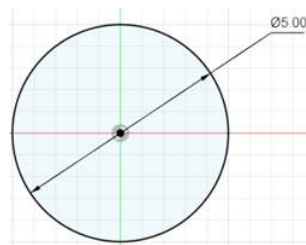


Figure 1.6.13: Pin - Step 1.

1.6.4.2 Step 2 - Extrude circle

Extrude the circle created in the previous step. The extrude length is 1 mm. (See Figure 1.6.14.)

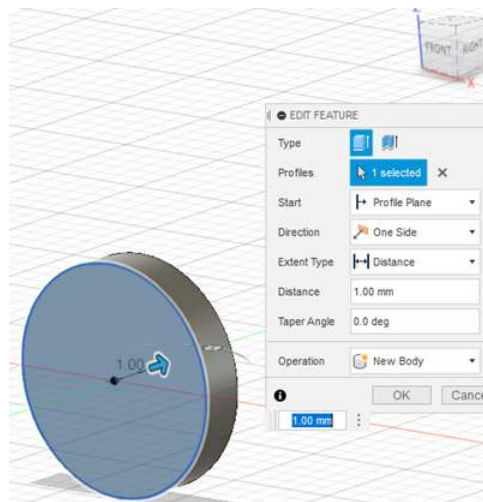


Figure 1.6.14: Pin - Step 2.

1.6.4.3 Step 3 - Sketch concentric circle

Sketch a concentric circle of diameter 2.5 mm on the face of the circular disc created in the previous step. (See Figure 1.6.15.)

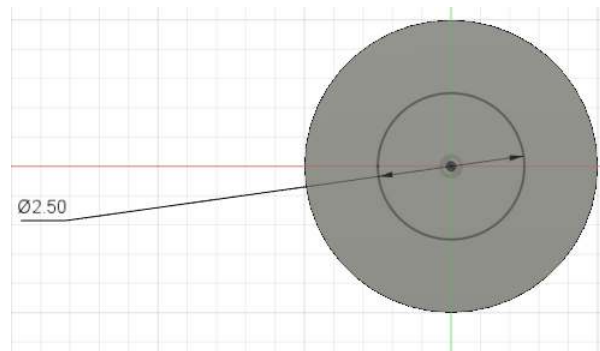


Figure 1.6.15: Pin Step 3.

1.6.4.4 Step 4 - Extrude circle

Extrude the circle created in the previous step. The extrude length is 10 mm. (See Figure 1.6.16.)

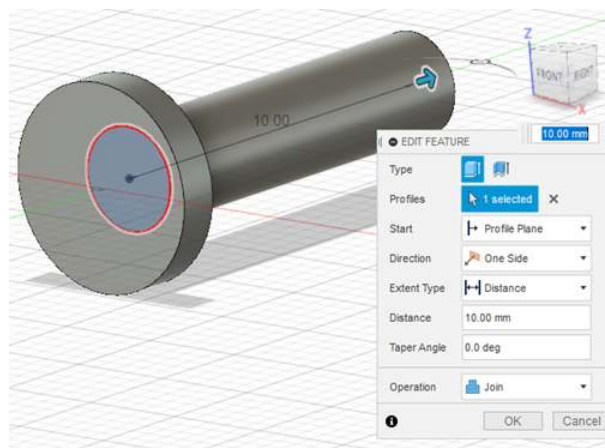


Figure 1.6.16: Pin Step 4.

1.6.4.5 Step 5 - Save file

Save the completed Pin created in the previous steps. (See Figure 1.6.17.)



Figure 1.6.17: Pin Step 5 - Save As dialog box.

1.6.5 Create the Final Assembly

In order to create the final assembly of all the components created in the previous steps, they have to be imported into an assembly file and then all the joint constraints need to be applied.

1.6.5.1 Step 1 - Create and save new design

Create a New Design. Save this New Design.

1.6.5.2 Step 2 - Insert base plate into assembly

Click on the Hide Data Panel to reveal the saved components saved in the account's cloud library. Right click on the Base Plate and select "Insert into Current Design". The Screen should look like the following, Figure 1.6.18. Then, select "OK" from the Move/Copy dialogue box. The base plate should turn into a gray color. This implies the position of the component is now fixed.

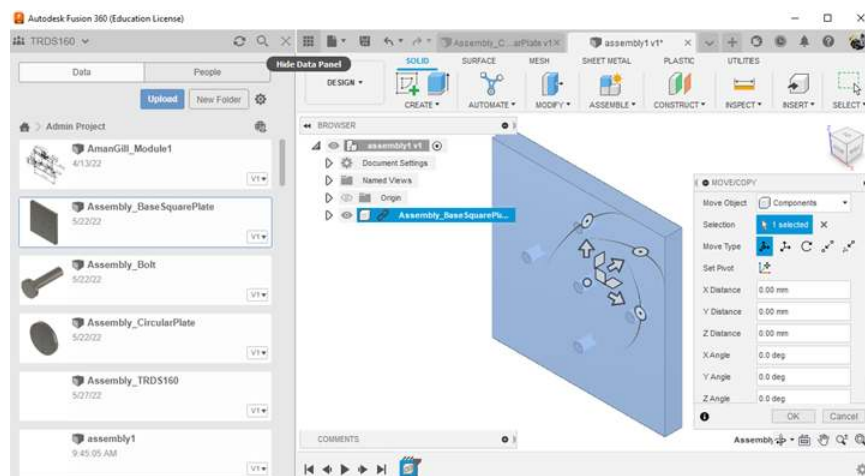


Figure 1.6.18: Assembly - Step 2 - Adding the Base Square Plate, using the Data Panel.

1.6.5.3 Step 3 - Import Circular plate

Similarly, import the circular plate into the design. Then select the Joint option from the Assemble drop down menu. Select the origin of the center hole on the bottom of the circular plate as shown in Figure 1.6.19.

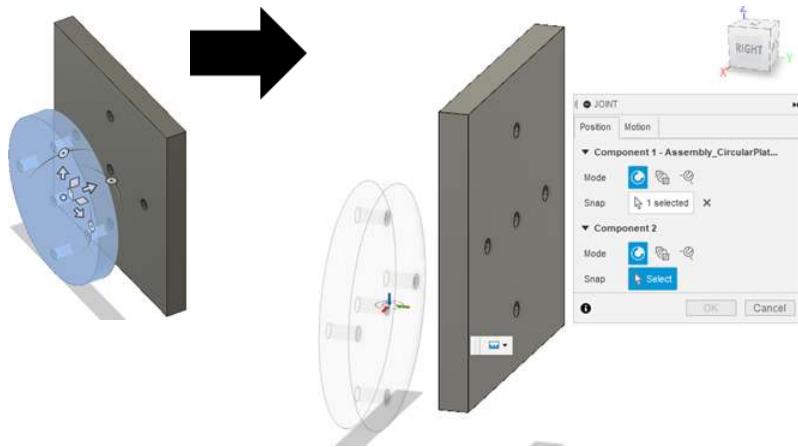


Figure 1.6.19: Step 3 - Importing Circular Plate and Selecting Center Hole origin.

1.6.5.4 Step 4 - Create joint

Next, select the origin on the center hole on the front face of the base plate. The circular plate will sit flush on the base plate as shown in Figure 1.6.20. All the holes on the plates will also be aligned after this step is executed correctly.

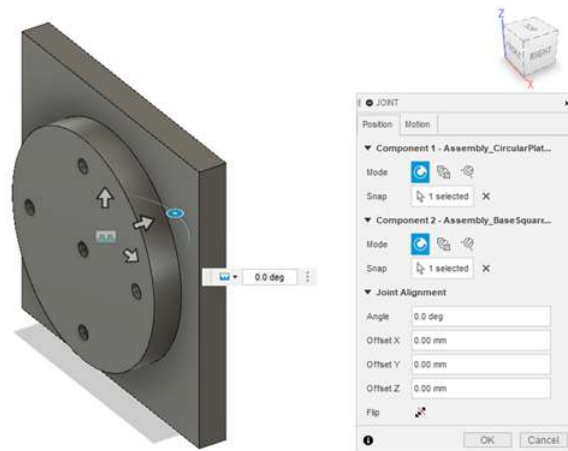


Figure 1.6.20: Step 4 - Holes of the Circular and Square base plates aligned, plates sitting flush.

1.6.5.5 Step 5 - Import pin

Import a Pin into the current assembly. Then select Joint from the Assembly drop down menu. Select the center of the bottom of the flat head and then select the hole you want to place the rivet in. The Figure 1.6.21 below shows this step.

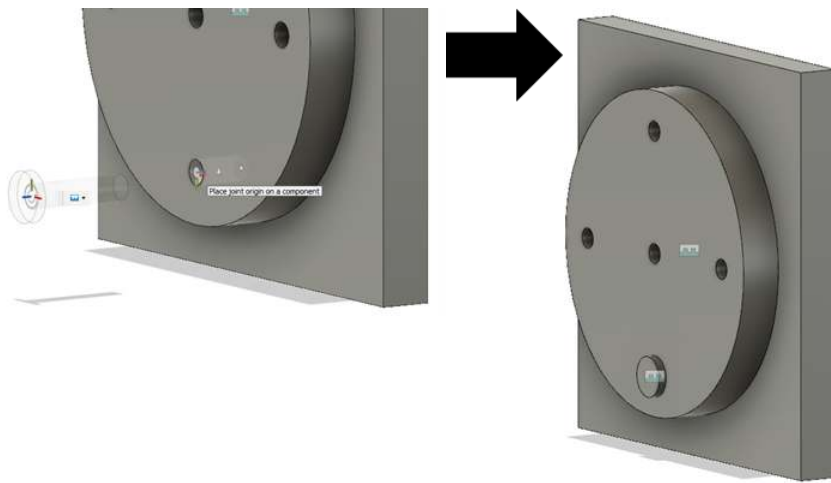


Figure 1.6.21: Step 5 - Aligning and setting pin in hole in plates.

1.6.5.6 Step 6 - Set remaining pins

Repeat Step 5 four times to set pins in the remaining four (4) holes. Save the completed assembly. (See Figure 1.6.22.)



Figure 1.6.22: Completed Assembly.

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1.7: Module 6 - Advanced Assembly

1.7.1 Introduction

Autodesk Fusion 360 allows for the assembly of multiple 3D components to form a new system of components. This system is referred to as an assembly or subassembly.

As an example, we will assemble a hinge assembly shown in Figure 1.7.1.

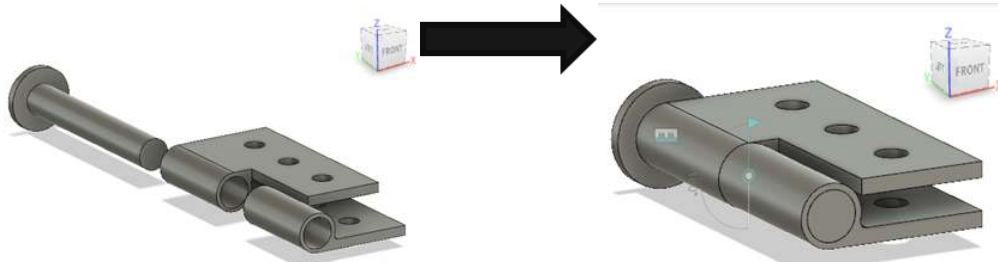


Figure 1.7.1: Objective assembly for this module.

1.7.1.1 Assembly Options

The assembly options discussed in the sub-sections below have already been discussed in the previous module but have been reiterated here for the sake of ease of reference.

To assemble various components together, there are a few options available. Some of the more frequently used are discussed in the sections below and are shown in Figure 1.7.2.

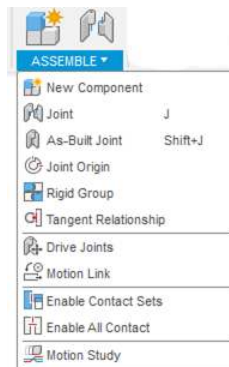


Figure 1.7.2: Assembly options in Fusion 360 - Assemble dropdown menu.

1.7.1.1.1 New Component

The New Component tool allows for adding a component or creating a new component that will be used in the assembly.

1.7.1.1.2 Joint

The Joint tool allows for adding various components relative to each other in the assembly and also specify relative motion between these components.

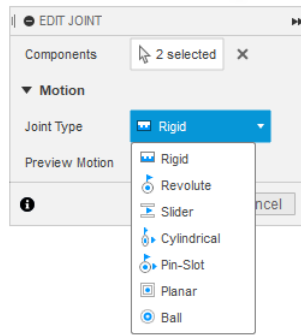


Figure 1.7.3: Sub-options for the assembly option Joint.

Joint Type options for the Joint are as follows:

- **Rigid** locks all components together and removes all degrees of freedom.
- **Revolute** rotates the component around the joint origin.
- **Slides** allows for sliding a component along an axis.
- **Cylindrical** rotates and slides a component on a single axis.
- **Pin Slot** rotates and slides the component on two different axes.
- **Planar** rotates a component around one axis normal to a plane and slides it along two axes parallel to the plane.
- **Ball** rotates a component on all three axes on a gimbal system.

The “rigid” option has been used in the previous module, so in this module the emphasis will be on creating a revolute joint and animating the joint.

1.7.1.1.3 As-Built Joint

The As-Built Joint tool positions components next to each other and defines the relative motion. The components maintain their original position.

1.7.1.1.4 Joint Origin

The Joint Origin tool positions the joint origin on the component. Joint origins defines the geometry used to relate a joint’s components.

1.7.1.1.5 Rigid Group

The Rigid Group tool locks the relative position of the selected components. Once this is applied, the components act as a single component.

1.7.1.1.6 Drive Joints

The Drive Joints tool specifies the rotation angle or distance value for the joint or degrees of freedom. This option usually works in conjunction with Joint assembly options.

1.7.2 Module 6 step by step video:



Computer Aided Design for Industry

Module 6 : Advanced Assembly

<https://youtu.be/as3hmYTwna0> - Transcript available on YouTube.

1.7.3 Create the Hinge

1.7.3.1 Step 1: Sketch

Create the sketch on the front plane for the hinge as shown in the image below. Use the skills learnt in the sketching module to accomplish this. All units are in millimeters (mm). The thickness of the plate is 2.50 mm the length of the flat portion of the base plate is 25 mm, the diameter of the outer circle is 10 mm, and the diameter of the inner circle is 8 mm. The radius of the fillet between the outer circle is 1 mm. Use appropriate constraints to make sure the sketch is fully constrained. (See Figure 1.7.4.)

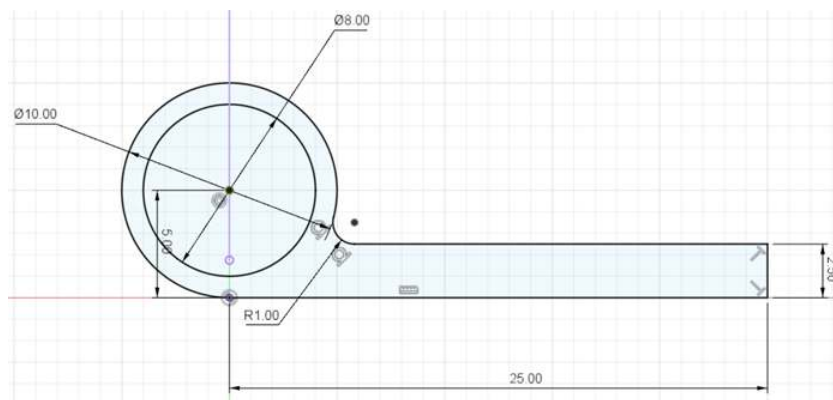


Figure 1.7.4: Step 1 - Sketch front plane of hinge.

1.7.3.2 Step 2: Extrude

Once the sketch is completed, use the extrude option to extrude the sketch to a length is 25 mm. (See Figure 1.7.5.)

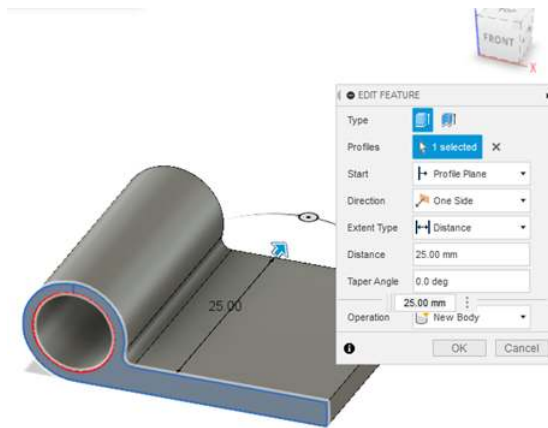


Figure 1.7.5: Step 2 - Extrude hinge plate.

1.7.3.3 Step 3: Sketch

Sketch a rectangle on the rectangular part of the base extruded base plate. The length of this rectangle is 20 mm and the width is 2.50 mm. (See Figure 1.7.6.)

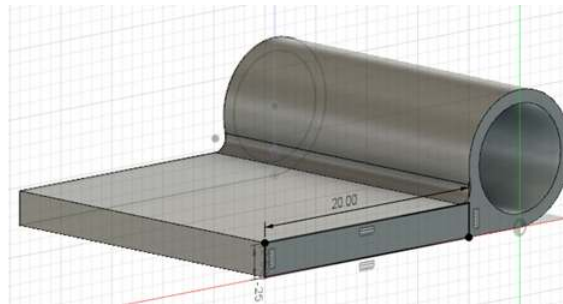


Figure 1.7.6: Step 3 - Sketch a rectangle.

1.7.3.4 Step 4: Extrude

Extrude the rectangle sketched in the previous step. The extrude length is 25 mm. (See Figure 1.7.7.)

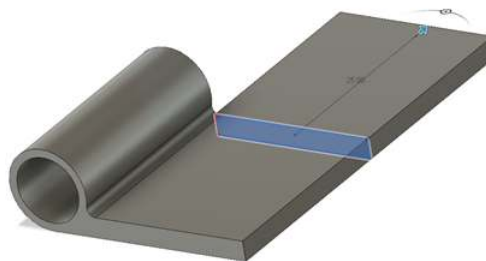


Figure 1.7.7: Step 4 - Extrude rectangle to create the remainder of the hinge plate.

1.7.3.5 Step 5: Sketch

Sketch three circles of diameter 5 mm on the top face of the hinge plate as shown in the figure 1.7.8 below. Each circle is colinear and situated 15 mm away from the previous one and 10 mm away from the shorter edge of the plate and 9.59 mm away from the longer edge of the plate.



A technical drawing of a circle on a grid. A horizontal red line passes through the center of the circle, representing a diameter. A green vertical line also passes through the center. A dimension line with arrows at both ends is drawn above the circle, parallel to the horizontal red line. The dimension line is labeled with the text "Ø16.00".

<https://workforce.libretexts.org/@go/page/25072>

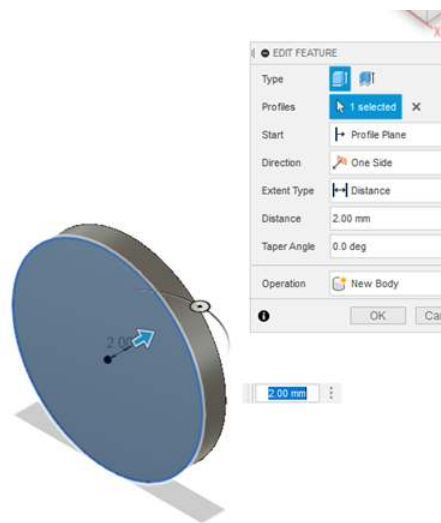


Figure 1.7.11: Step 2 - Extrude circle.

1.7.4.3 Step 3: Sketch

Sketch a concentric circle of diameter 8 mm on the face of the circular disc created in the previous step. (Figure 1.7.12.)

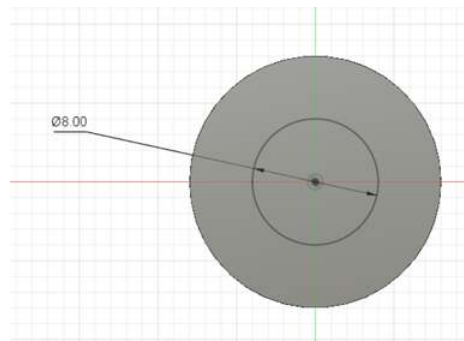


Figure 1.7.12: Sketch concentric circle.

1.7.4.4 Step 4: Extrude

Extrude the circle created in the previous step. The extrude length is 50 mm. (Figure 1.7.13.)

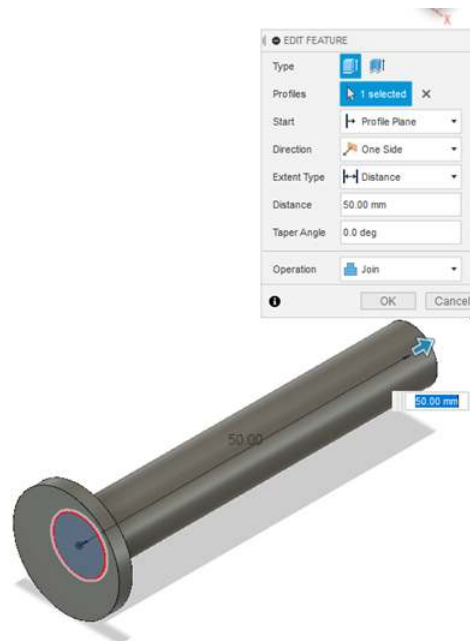


Figure 1.7.13: Step 4 - Extrude the circle.

1.7.4.5 Step 5

Save the completed hinge pin created in the previous steps.

1.7.5 Create the Final Assembly

In order to create the final assembly of all the components created in the previous steps, they have to be imported into an assembly file and then all the joint constraints need to be applied.

1.7.5.1 Step 1

Create a New Design. Save this New Design.

1.7.5.2 Step 2

Click on the Bento box menu to reveal the saved components saved in the account's cloud library.

1.7.5.3 Step 3

Right click on the Hinge and select "Insert into Current Design". Do it one more time. The workspace should now have two Hinges and the screen should look like Figure 1.7.14. Then, select "OK" from the Move/Copy dialogue box. The base plate should turn into a gray color. This implies the position of the component is now fixed. (See Figure 1.7.14.)

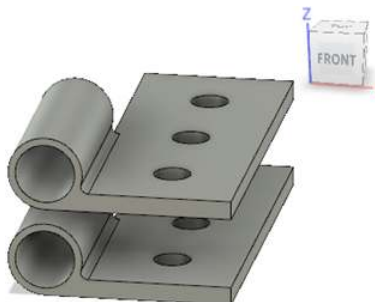


Figure 1.7.14: Step 3 - Insert two hinge plates into the assembly.

1.7.5.4 Step 4

Create a revolute joint between the two hinges. The joint can only rotate 180° and there is no offset in the X, Y, or Z axis. The new joint should look like the following Figure 1.7.15.

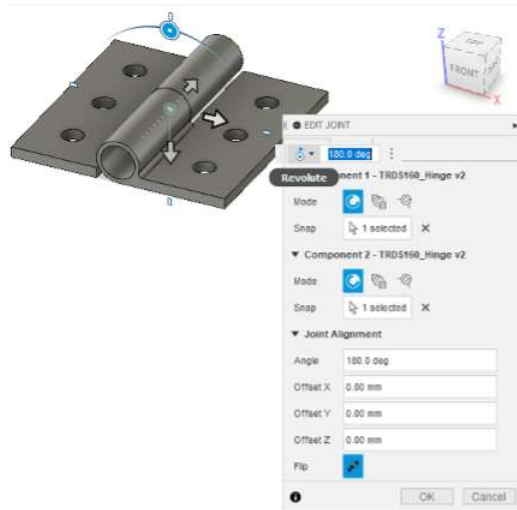


Figure 1.7.15: Step 4 - create revolute joint between two pieces.

1.7.5.5 Step 5

Right click on the Hinge Pin and select “Insert into Current Design”. (See Figure 1.7.16.)

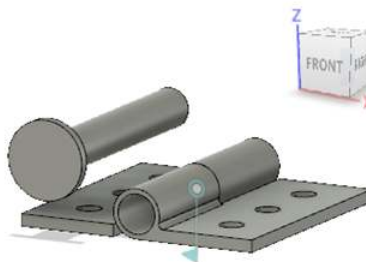


Figure 1.7.16: Step 5 - Insert Hinge Pin into assembly.

1.7.5.6 Step 6

Create a rigid joint to make sure the Hinge Pin is fixed through both the hinges. The underside of the Hinge Pin should be flush with the top surface of the hinge hole. There is no angle or offset in any axis for this joint. (See Figure 1.7.17.)

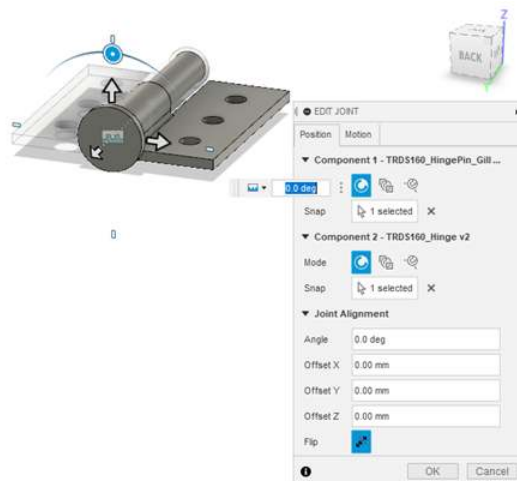


Figure 1.7.17: Create rigid joint between hinge pin and hinge pieces.

1.7.5.7 Step 7

Save the assembly.

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1.8: Module 7 - Drawings

1.8.1 Introduction

This module looks at creating engineering drawings from a three-dimensional model. Engineering drawings are essentially a communication tool. They are typically used by the designer to communicate with:

- a. other designers,
- b. machinists,
- c. quality technicians, and
- d. supply chain associates.

Projecting a three-dimensional object on to two dimensions presents its own challenges. Multiple views of the three-dimensional entity must be created in order to supply complete information in a two-dimensional medium. This two-dimensional medium in this case is the computer screen. However, the drawing space is chosen according to the size of the paper it will eventually be printed on. Paper sizes are classified as American Society of Mechanical Engineers (ASME) A – E, with A being the smallest paper size and E being the largest paper size. Other classifications such as Organisation Internationale de Normalisation (ISO) A0 – A4 can also be used, with A0 being the largest paper size, and A4 being the smallest paper size).

1.8.2 Two-Dimensional Views of Three-Dimensional Objects

The two primary ways to represent a three-dimensional object in two dimensions are: Orthogonal Views and Auxiliary Views. However, three-dimensional objects can also be rendered in a way to give them some sense of depth. This happens in real time in the Autodesk Fusion 360 environment. To translate this three-dimensional rendering on to a two-dimensional drawing, a technique called the isometric view, as shown in Figure 1.8.1, is used. Note that for a highly symmetrical object with no angled surfaces, the isometric view works well to convey all the information about the dimensions of the object.

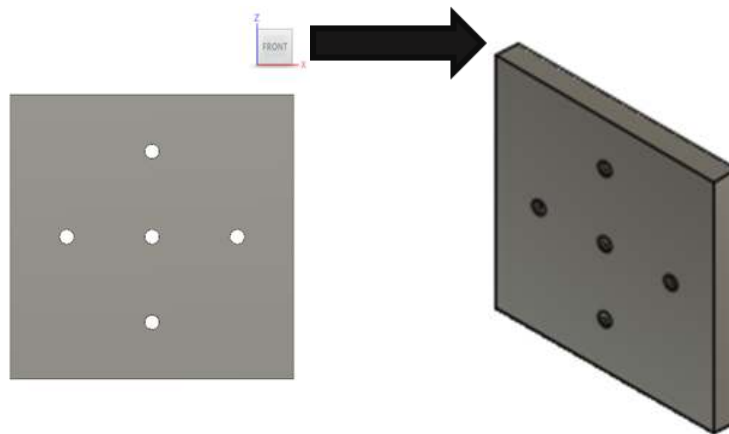


Figure 1.8.1: Isometric view (left) of a three-dimensional rendering (right) of the square base plate covered in Module 5.

1.8.2.1 Orthogonal View

To perceive orthogonal views, imagine the object being freely suspended inside a glass box. To look at the object you could place it such that you are looking straight at it in one of six orientations: top/bottom, front/back, and left/right side. All of these six views have been shown in Figure 1.8.2.

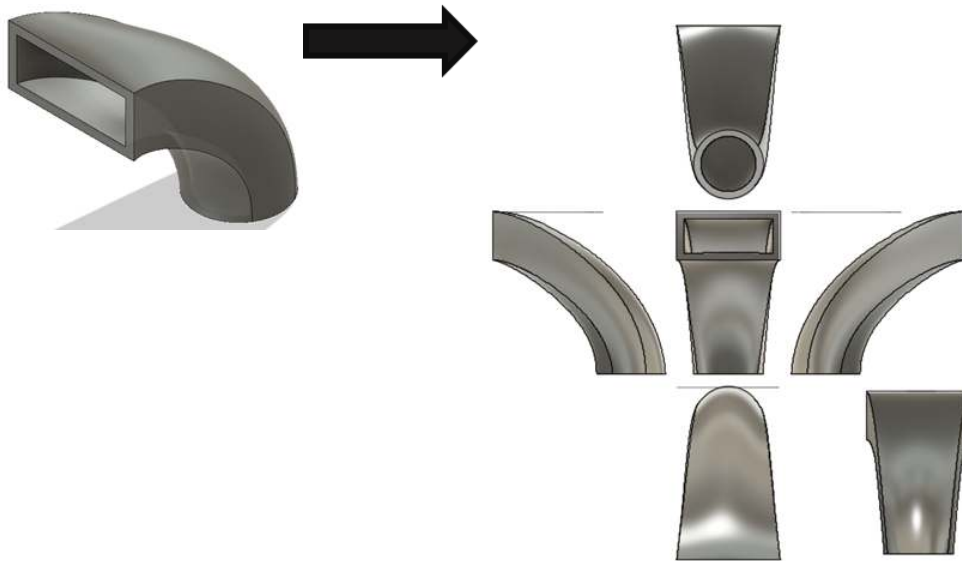


Figure 1.8.2: Orthogonal views for a Heating Ventilation and Air Conditioning (HVAC) duct.

1.8.2.2 Auxiliary View

Any view obtained by a projection on a plane other than the horizontal (H), frontal (F) and profile (P) is an auxiliary view.

For a three-dimensional object with angled surfaces that cannot be projected accurately on the planes mentioned above, there need to be auxiliary planes that will accomplish that. In Figure 1.8.3, when the orthogonal view of the bracket is created, the length of the angled side is measured at 40.41 millimeters (mm). However, this measurement is inaccurate. In order to get the accurate measurement on the drawing, it must be measured on an auxiliary plane parallel to the angled edge.

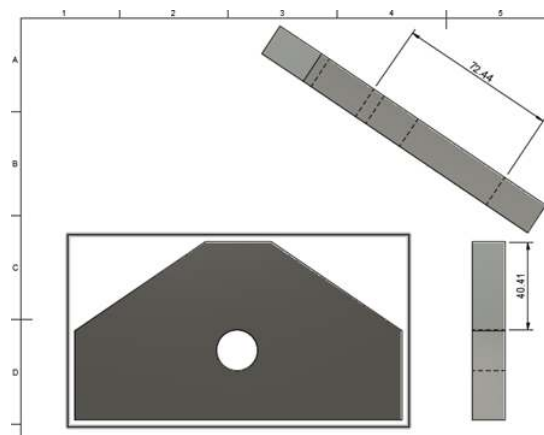


Figure 1.8.3: Projection on Auxiliary Planes.

1.8.3 Dimensioning

As noted in the previous sections, accurate dimensions are an essential component of the engineering drawing. There are several strategies to place dimensions in an engineering drawing. Dimensions can be placed on the X or Y axis, can be continuous or broken down, or can be placed such that they align with the shape being labeled. Text can be added to engineering drawings in order to add any specific notes or instructions. Like dimensions, text can be customized to be placed in different orientations, fonts, and alignments. All the available options for dimensioning in Autodesk Fusion 360 are shown in Figure 1.8.4.

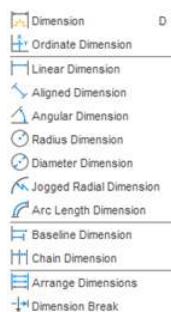


Figure 1.8.4: Dimensioning options available in the drawing mode of Autodesk Fusion 360.

1.8.3.1 Dimension

The Dimension option allows for creating linear, aligned, angular, radius, and diameter dimensions for a drawing entity, such as a line, arc, circle or points. This option automatically latches on to the type of the entity. (See Figure 1.8.5)



Figure 1.8.5: Icon for the Dimension option.

1.8.3.2 Ordinate Dimension

The Ordinate Dimension option allows for creating dimensions references from an origin defined by the user. (See Figure 1.8.6.)



Figure 1.8.6: Icon for the Ordinate Dimension option.

This option is especially useful if the component in the drawing has multiple dimensions along the X or Y axis that will be useful during manufacturing the component. For example the hinge shown in Figure 1.8.7, has the center points of all the holes on the plate of the hinge dimensioned using ordinate dimensions. If the hole were to be drilled out using a drill press, the drill operator would not need to do any additional calculations to see how far the drill head or the workpiece must be moved from the base of the hinge plate to accurately drill the holes.

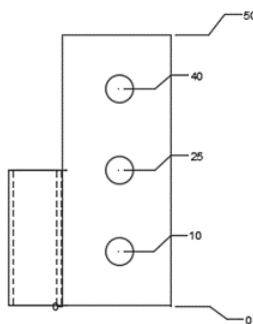


Figure 1.8.7: An example of Ordinate Dimensioning used for a hinge base plate.

1.8.3.3 Linear Dimension:

The Linear Dimension option allows for measuring the horizontal or vertical distance between two points or an edge. (See Figure 1.8.8.)



Figure 1.8.8: Icon for the Linear Dimension option.

1.8.3.4 Aligned Dimension

The Aligned dimension option allows for measuring the distance between two points that may not be either vertical or horizontal. (See Figure 1.8.9.)



Figure 1.8.9: Icon for the Aligned Dimension option.

Figure 1.8.10 shows the hexagonal head of an M7 bolt. Since it is a regular hexagon, all its sides are the same length. To label the length of a horizontal edge of the hexagon, linear dimension can be used. However, if the user chooses to dimension one of the non-horizontal edges, then an aligned dimension should be used.

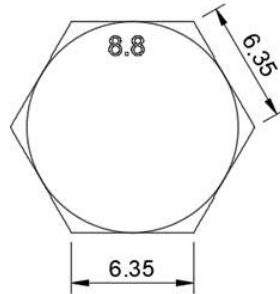


Figure 1.8.10: Linear and Aligned Dimension used for the hexagonal head of an M7 bolt.

1.8.3.5 Radius Dimension

The Radius dimension option allows for dimensioning a circular entity using its radius. (See Figure 1.8.11.)



Figure 1.8.11: Icon for Radius Dimension option.

1.8.3.6 Diameter Dimension

The diameter dimension option allows for dimensioning a circular entity using its diameter. (See Figure 1.8.12.)



Figure 1.8.12: Icon for the Diameter Dimension option.

Consider the front view of the hinge base plate as shown in Figure 1.8.13. The inner and outer diameter of the hole where the pin is supposed to sit is dimensioned with the diameter annotation ϕ (Greek alphabet letter phi). The fillet radius is dimensioned with radius annotation R.

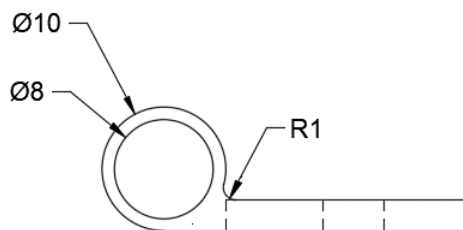


Figure 1.8.13: Circular features on the front view of a hinge base plate dimensioned using Radius and Diameter Dimension.

1.8.3.7 Jogged Radial Dimension

The Jogged Radial dimension allows for dimensioning a circular component with dimension running from the center of the circular entity to its edge. It can also be jogged to a more convenient place within the circular entity for increased legibility. (See Figures 1.8.14 and 1.8.15.)



Figure 1.8.14: Icon for the Jogged Radial Dimension option.

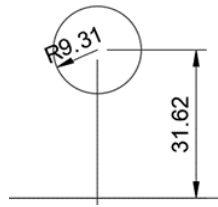


Figure 1.8.15: Example of a jogged radial dimension placed away from the linear dimension.

1.8.3.8 Arc Length Dimension

The Arc Length dimension allows for dimensioning the length of an arc. (See Figure 1.8.16.)



Figure 1.8.16: Icon for the Arc Length option.

Consider the C-Clamp of a vice shown in Figure 1.8.17. There are several arcs used in solid model. They can be labeled in the drawing using the arc length option.

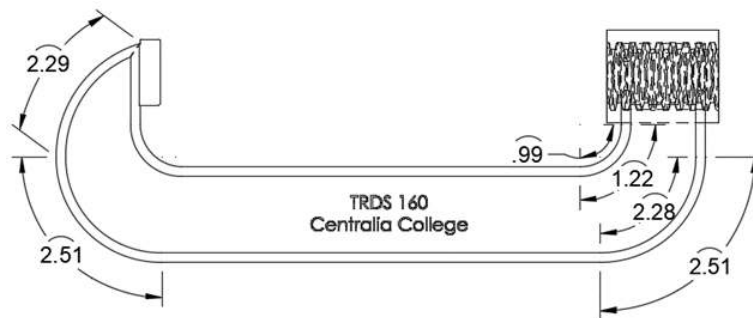


Figure 1.8.17: Arc Length Dimension option used to label the arc lengths of a C clamp.

1.8.3.9 Baseline Dimension

The Baseline Dimension option allows the user to use an existing linear dimension as a base point to create a new linear dimension. (See Figures 1.8.18 and 1.8.19.)



Figure 1.8.18: Icon for the Baseline Dimension option.

The holes of the hinge base plate shown in Figure 1.8.19 have been dimensioned using baseline dimension option. The base dimension used is the distance from the base to center of the first hole, i.e. 10 mm.

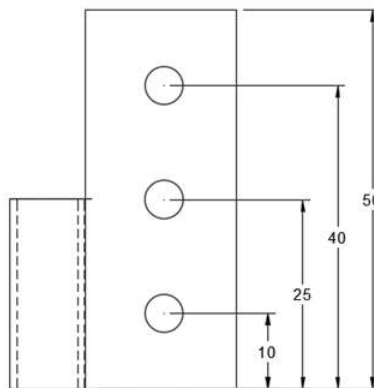


Figure 1.8.19: Baseline Dimension option used to label the distances of the holes from the base of the hinge plate.

1.8.3.10 Chain Dimension

Allows the user to use an existing linear dimension to create a new linear chain dimension.



Figure 1.8.20: Icon for the Chain Dimension option.

The holes of the hinge base plate shown in Figure 1.8.21 have been dimensioned using chain dimension option. The base dimension used is the distance from the base to center of the first hole, i.e. 10 mm.

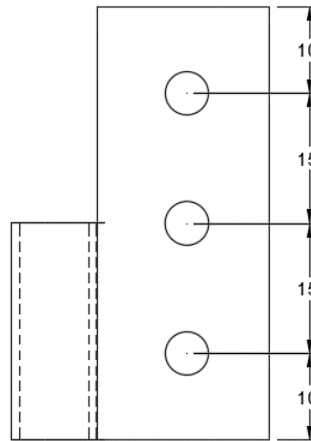


Figure 1.8.21: Chain Dimension option used to label the distances of the holes from the base of the hinge plate.

1.8.4 Geometric Dimensioning and Tolerancing

Geometric Dimensioning and Tolerancing (GD&T) is used in the industry to communicate dimensioning specifications using a standard set of symbols. These symbols are reviewed and prescribed by ASME in their Y14.5-2018 standard and internationally by ISO in their 1101:2017 standard. Fusion 360 incorporates most of these symbols under its symbols option as shown in Figure 1.8.22.



Figure 1.8.22: Icons for the GD&T options available in the drawing mode of Autodesk Fusion 360.

1.8.4.1 Surface Finish

The surface texture symbol allows for the designer to specify the machining operations, minimum and maximum surface roughness, and the amount of material that is to be removed from a surface. The surface finish symbol is shown in Figure 1.8.23. The dialog box for inputting the surface finish parameters is shown in Figure 1.8.24. To apply the surface finish symbol to a solid model imported into a drawing, select an edge or feature to which the symbol has to be applied. In Figure 1.8.24, the base plate of the hinge has a surface finish symbol applied to it. The operation specified is milling, the maximum and the minimum surface roughness is 3 and 2 micrometers, machining allowance is 0.01 mm, and the sampling length is 25 mm. The symbol is generated and applied as per ASME standard.



Figure 1.8.23: Icon for the Surface Finish option.

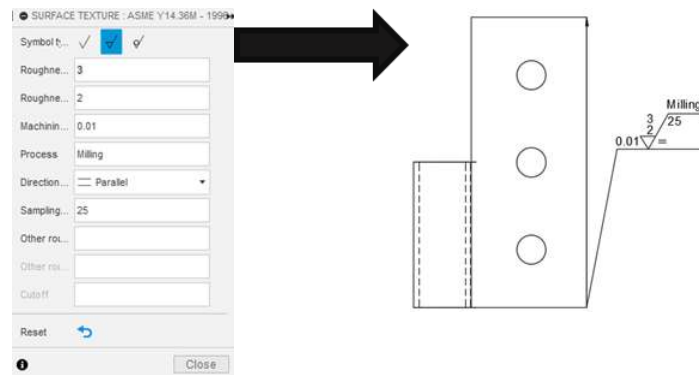


Figure 1.8.24: Surface Finish Symbols as applied to the base plate of the hinge.

1.8.4.2 Feature Control Frame

The feature control frame allows the user to call out a specific feature of the solid model in a drawing and add symbols that signify the tolerances for it. The icon for feature control frame is shown in Figure 1.8.25. The dialog box for inputting the feature control parameters is shown in Figure 1.8.26. To apply the surface finish symbol to a solid model imported into a drawing, select a feature to which the symbol has to be applied. In Figure 1.8.26, a hole on the base plate of the hinge has a control frame symbol applied to it. The tolerance specified is circular runout, which cannot exceed 0.01 mm around the datum A, which is the axis of the hole feature. The symbol is generated and applied as per ASME standard.

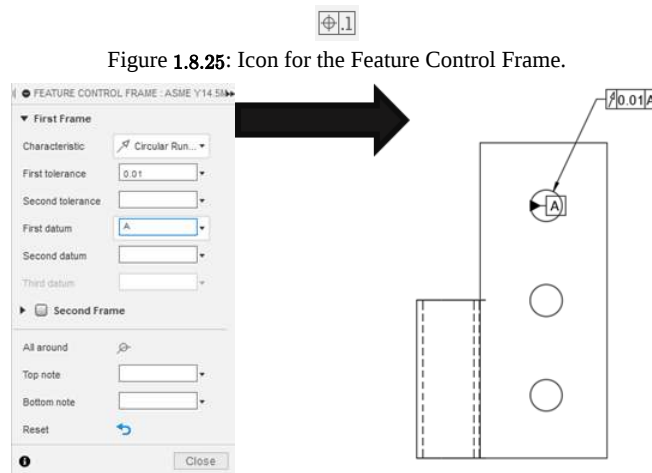


Figure 1.8.25: Icon for the Feature Control Frame.

Figure 1.8.26: Feature Control Frame as applied to the base plate of the hinge.

1.8.4.3 Other symbols

In addition to surface texture and feature control, Fusion 360 also allows for other symbols associated with GD&T, such as datum identifiers, taper and slope identifiers and welding symbols. These symbols are also reviewed and prescribed by ASME in the United States and internationally by ISO.

1.8.5 Create a drawing from a Solid model

First a sketch of the solid model will have to be created. Once the sketch has been completed, the solid model will be extruded and a hole feature would be added. Then, the solid model will be imported into a drawing.

1.8.5.1 Module 7 step-by-step videos:



Computer Aided Design for Industry

Module 7 : Drawings

Part 1 - <https://youtu.be/QyLWUj-JMSs> - Transcript is available on YouTube.



Computer Aided Design for Industry

Module 7 : Drawings

Part 2 - https://youtu.be/0pvv_LE_wj8 - Transcript is available on YouTube.

1.8.5.2 Step 1 - Create the sketch

Start a new sketch file. Create and constrain the sketch as shown in the Figure 1.8.27. It is a six-sided polygon with base length of 150.042 mm, sides of 40.839 mm, angled sides of 72.44 mm, and top edge 29.936 mm. Make sure to finish the sketch after completion.

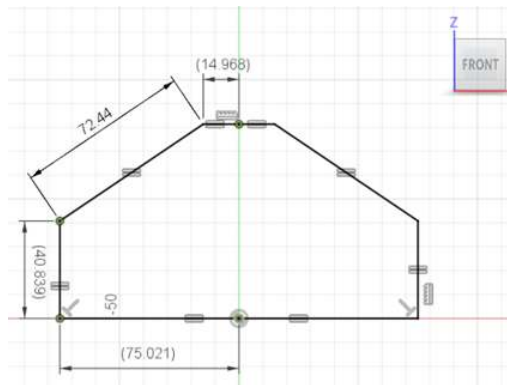


Figure 1.8.27: Sketch of the solid model to be created.

1.8.5.3 Step 2 - Extrude the sketch

Extrude the sketch to a thickness of 15 mm as shown in the figure 1.8.28.

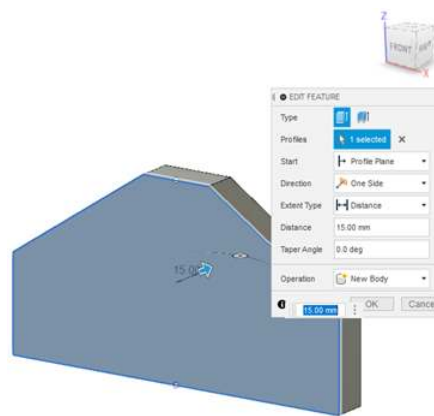


Figure 1.8.28: Extruded sketch.

1.8.5.4 Step 3 - Sketch a circle on the face of the solid model

Sketch a hole on the face of the solid model just created. Use the dimensions and the centerline to create the circle. The center of the circle of diameter 18.623 mm is on the centerline and is 49.726 mm away from the top edge as shown in Figure 1.8.29.

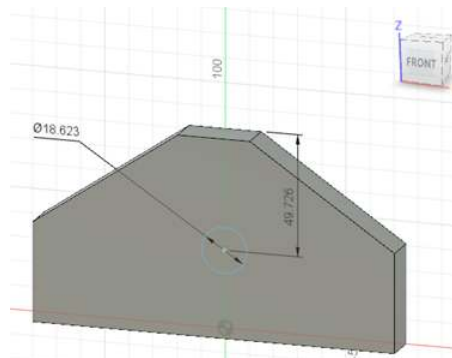


Figure 1.8.29: Sketch a circle on the face of the solid model.

1.8.6 Step 4 - Extrude cut the circle to form a hole

Extrude cut the circle sketch to form a through hole as shown in 1.8.30. This is a one sided through hole. The finished model will look like the one in figure 1.8.31.

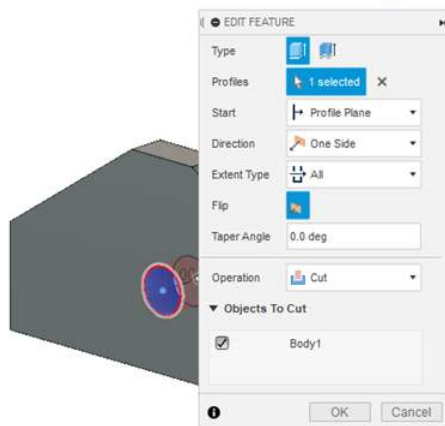


Figure 1.8.30: Extrude cut hole in solid model.

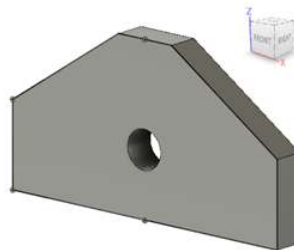


Figure 1.8.31: Finished solid model.

1.8.6.1 Step 5 - Import the model into the drawing

While this solid model tab is still open, from the top left corner of the screen, click on “Design”. From the drop down menu, select Drawing > From Design as shown in Figure 1.8.32. The solid model will turn light blue and a dialog box for the setting for the drawing will open up.

Once you click Ok on the dialog box, the drawing tab opens up as shown in Figure 1.8.33. Select an appropriate sheet size and scale. For this particular model, an A3 sheet size and a scale of 1:2.

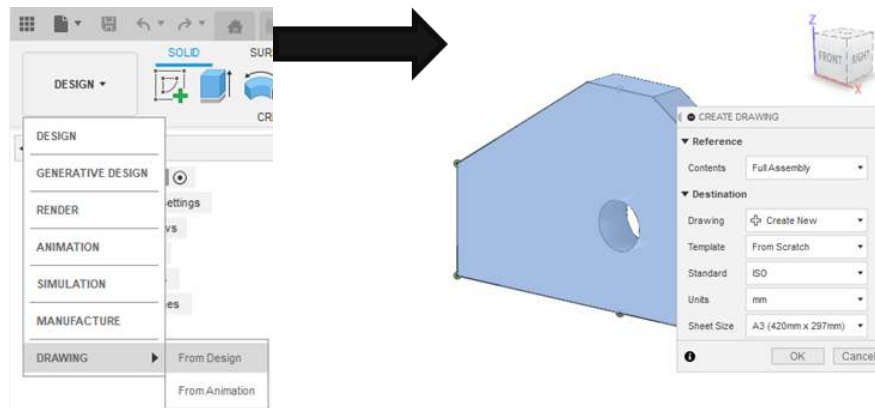


Figure 1.8.32: Importing a solid model into a drawing.

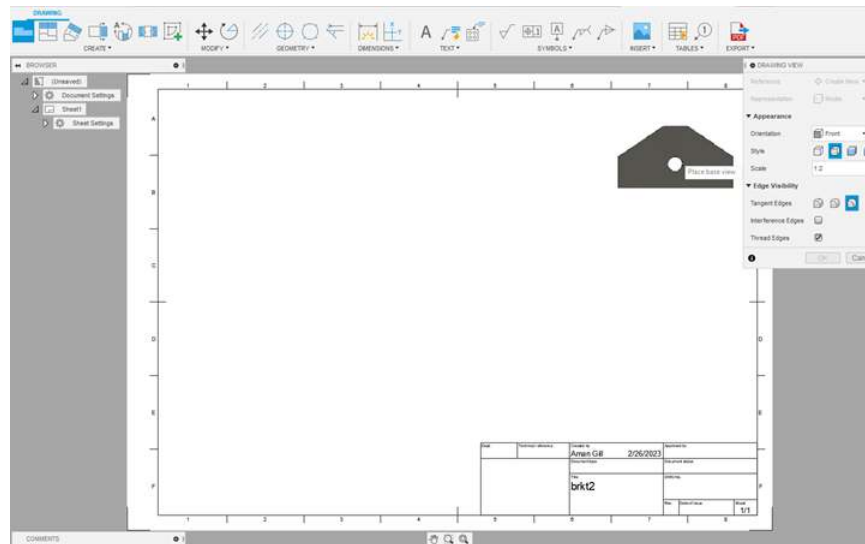


Figure 1.8.33: Drawing tab with the imported solid model.

1.8.6.2 Step 6 - Create additional views

Under the Create tab, additional views of the drawing can be created. As discussed earlier in this module, it is often insufficient to convey all the information involved in manufacturing the solid model from a 3D CAD file using just one view. All the options for adding these views are shown in 1.8.34.

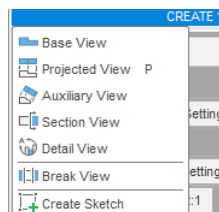


Figure 1.8.34: Additional views that can be added in the drawing.

1.8.6.2.1 Projected view

This option creates orthogonal and isometric views from the base view. A side view for this solid model can be added as shown in figure 1.8.35.

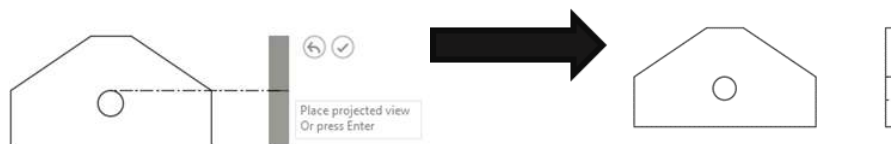


Figure 1.8.35: Adding a projected view for the solid model.

Similarly, a bottom view and an isometric view can be added. The resulting drawing should look like figure 1.8.36.

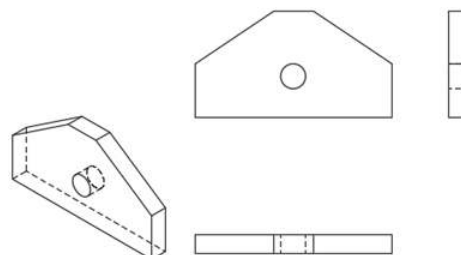


Figure 1.8.36: Base view and isometric view added using projected view.

1.8.6.2.2 Auxiliary View

Add an auxiliary view as shown in **1.8.37**. As discussed earlier in this module, auxiliary views are necessary for displaying the accurate dimensions of an angled feature on a solid model.

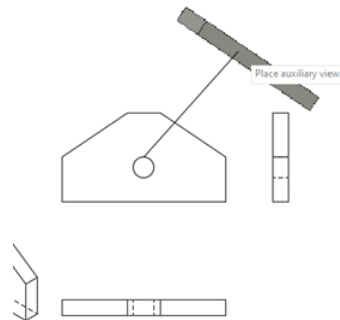


Figure 1.8.37: Adding an auxiliary view.

1.8.6.2.3 Section View

Select section view and place the sectioning datum as shown in figure **1.8.38**. Think of this datum as a knife that will slice this solid model in half. Once this datum, AA (scale 1:2), is finalized, place this section view on the left side of the base view. The hatching represents the solid sections of the solid model and the non-hatched part represents the hole in the center of this solid model.

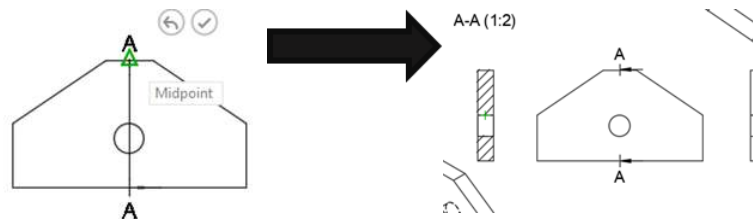


Figure 1.8.38: Adding a section view.

1.8.6.2.4 Detail View

In order to create a more detailed view of the hole, select the section view and select the section view as the base view as shown in figure **1.8.39**. A scale of 1:1 is appropriate for this.

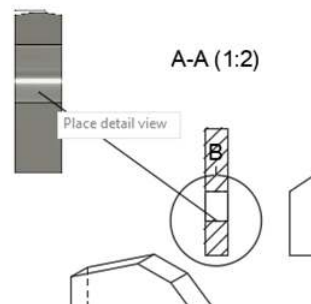


Figure 1.8.39: Adding a detail view.

The drawing sheet at this point should look like the figure **1.8.40**.

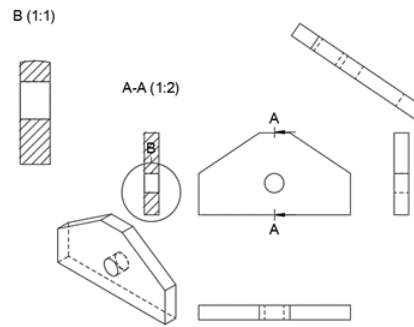


Figure 1.8.40: Drawing file after adding all the views.

1.8.6.2.5 Modifying the drawing

Should you choose to delete or move or rotate any of the views on the drawing sheet, the Modify drop down menu allows for this. All the options available are shown in figure 1.8.41.

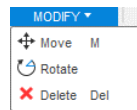


Figure 1.8.41: Modify options available to modify the views on the drawing sheet.

1.8.6.2.6 Geometry Call Outs

Should you choose to call out the geometry features on any of the views on the drawing sheet, the Geometry drop down menu allows for this. All the options available are shown in figure 1.8.42.

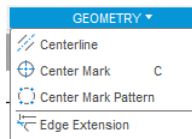


Figure 1.8.42: Geometry call out options.

1.8.6.3 Add Dimensions to the Drawing

As discussed in the previous sections in this module, add the following dimensions and symbols to the drawing as shown in figure 1.8.43.

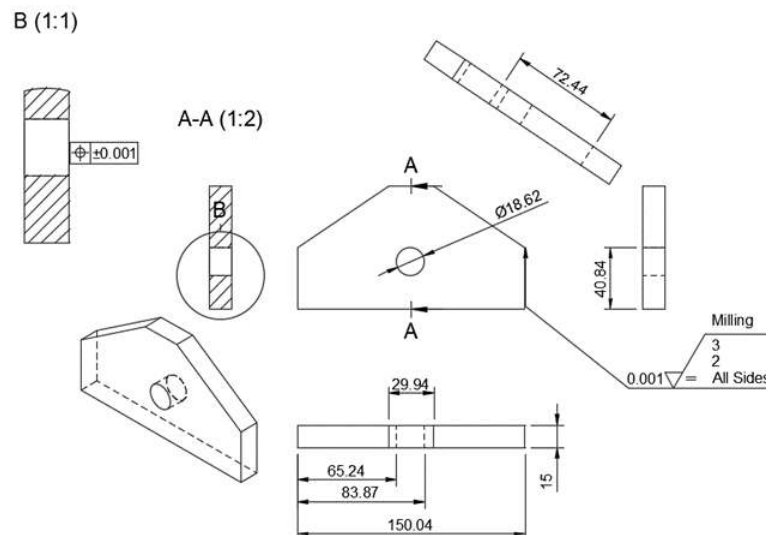


Figure 1.8.43: Completed drawing with all the dimensions and symbols.

1.8.7 Edit the Drawing Description

Double click the drawing description table and edit the description as appropriate. An example of this is shown in figure 1.8.44.

Dept. Centralia College	Technical reference Module: Drawing	Created by Aman Gill	2/26/2023	Approved by CC_CTE/Lib	2/27/2023
		Document type Class Demo	Document status Current		
		Title Bracket	DWG No. TRDS160M7R3.2		
		Rev. 3.2	Date of issue 2/28/2023	Sheet 1/1	

Figure 1.8.44: Drawing description.

1.8.7.1 Save and/or Export the Drawing

The drawing that has just been completed should now be saved. It can also be exported to a PDF file format using the export option as shown in figure 1.8.45.

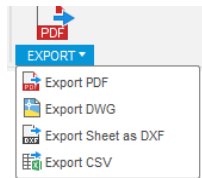


Figure 1.8.45: Export options to export drawing from an Autodesk Fusion 360 file to an Adobe Acrobat file (PDF).

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1.9: Final Project - Assembly

1.9.1 Introduction

A C-clamp is a cheap and portable alternative to a table vise. It is made of a piece of metal shaped like a 'C' and a bolt with a flat head. The bolt can go back and forth to clamp or unclamp a workpiece. The assembly of these two components forms a C-clamp.

1.9.2 Create the body of the C-clamp

In this section, the body of the C-clamp will be sketched and then then extruded.

1.9.2.1 Step 1 - Initial sketch of the clamp

Sketch the initial body as shown in Figure 1.9.1. Make sure the units are inches. Create the U shape using the Line option. The length of the line is 9.815 inches. Then round off the edges using the Fillet option. The radius of the fillet on each side is 1.597 inches. The inner line is offset at a distance of 1.45 inches. The fillet radius on this inner line is 0.629 inches.

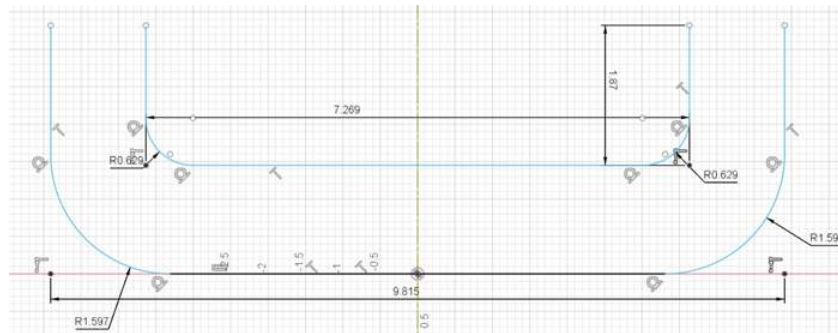


Figure 1.9.1: Initial Sketch.

Then, sketch a three-point arc on the left side of the initial sketch as shown in Figure 1.9.2. The radius of this arc is 1.803 inches.



Figure 1.9.2: Three-point arc on the left side of the sketch.

Next, use the Trim option to remove the extra bit of vertical line left over, as shown in Figure 1.9.3.

Finally, draw a horizontal line on the right side of the initial sketch to complete the sketch. The completed sketch is shown Figure 1.9.3.



Figure 1.9.3: Completed initial sketch.

1.9.2.2 Step 2 - Extrude the clamp

Extrude the sketch completed in the previous step. The extrude parameters are shown in the dialog box. The completed extrude and the parameters are shown in Figure 1.9.4.

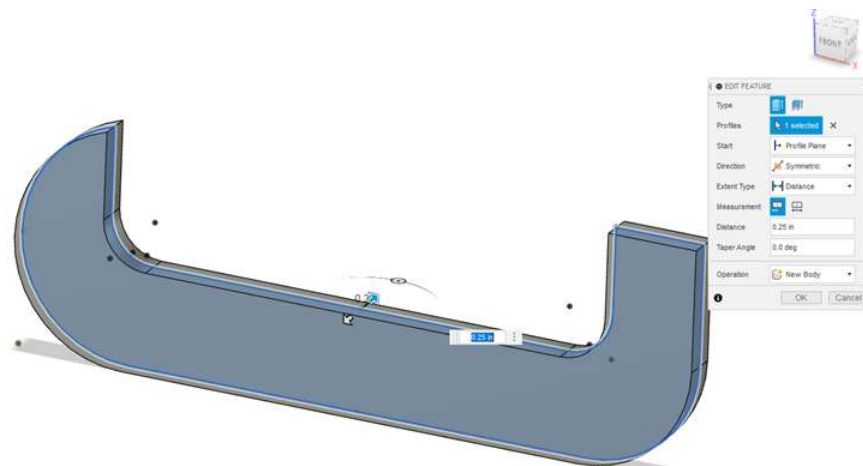


Figure 1.9.4: Extruded sketch.

1.9.2.3 Step 3 - Sketch clamping surface

Select the inner surface of the clamp and sketch a square of side 0.875 inches, with its center at 1.5 inches away from the edge of the clamp as shown in Figure 1.9.5.

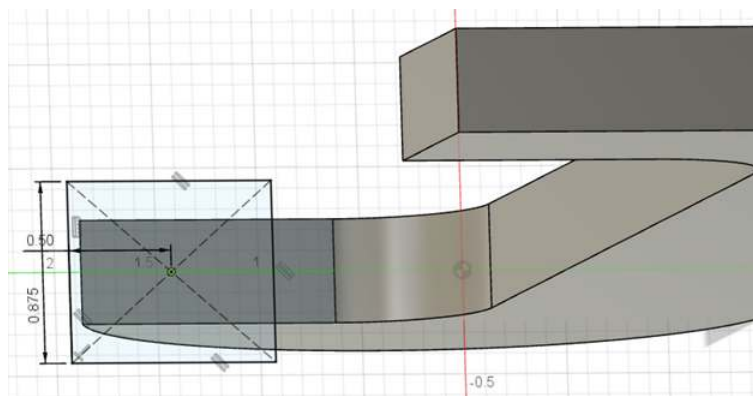


Figure 1.9.5: Sketch clamping surface.

1.9.2.4 Step 4 - Extrude the clamping surface

Extrude the sketch created in the previous step to a distance of 0.30 inches as shown in Figure 1.9.6.

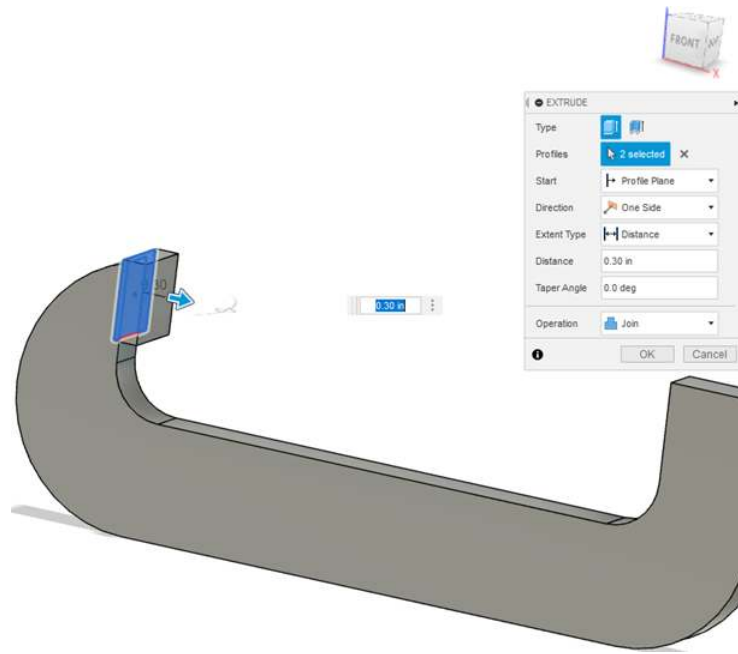


Figure 1.9.6: Extrude the clamping surface.

1.9.2.5 Step 5 - Create a lip on the edge of the clamp

Use the offset option and copy the existing edge of the clamp as shown in Figure 1.9.7. This offset sketch will be on the inside of the existing curve and at a distance of 0.146 inches as shown in Figure 1.9.8.

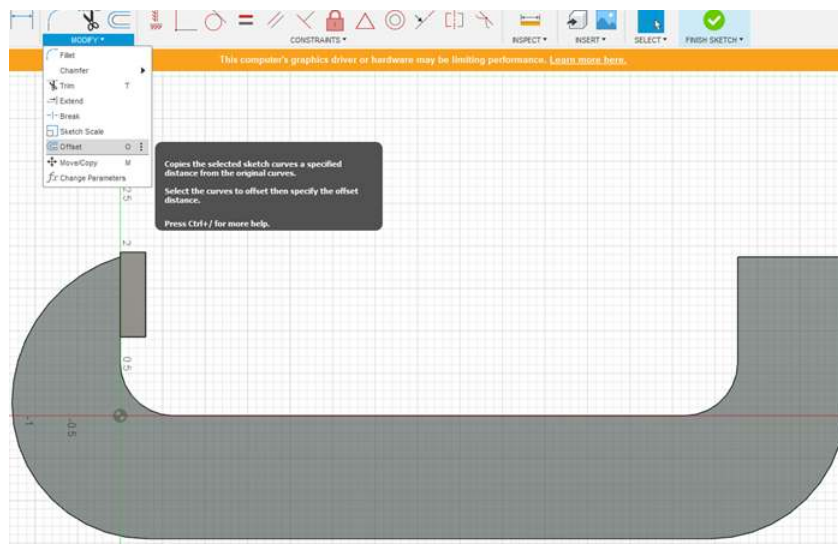


Figure 1.9.7: Use the Offset option to copy the create a sketch parallel to the pre-existing sketch of the clamp.

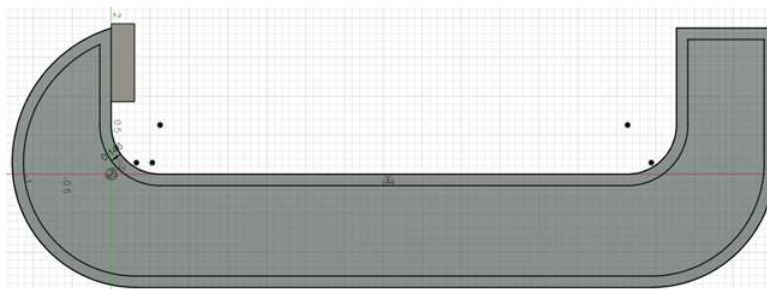


Figure 1.9.8: Sketch created using the Offset option.

1.9.2.6 Step 6 - Extrude the lip

Extrude the offset sketch created in the previous step to a distance of 0.20 inches. The extrude parameters are shown in Figure 1.9.9.

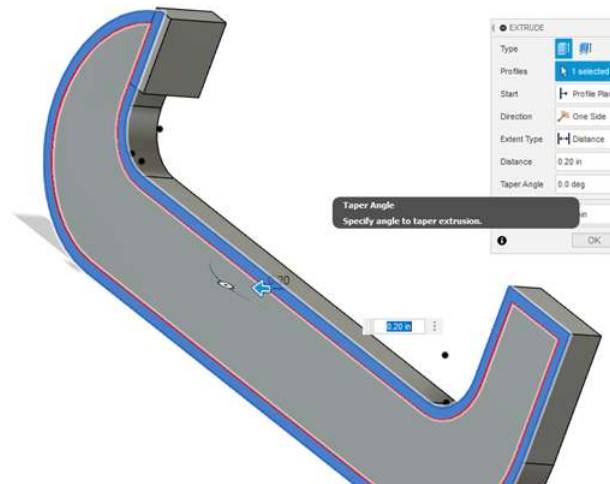


Figure 1.9.9: Extrude parameters for the lip.

1.9.2.7 Step 7 - Apply Fillet to the extruded lip

Apply a Fillet of 0.05 inches to the lip created in step 2.6. The end result of the applied fillets is shown in Figure 1.9.10.



Figure 1.9.10: Fillet applied to the extruded lip.

1.9.2.8 Step 8 - Apply Fillet to the edges of the clamping surface

Apply a Fillet of radius 0.020 inches to the edges of the clamping surface that was created in step 4. The result of this applied Fillet is shown in Figure 1.9.11.

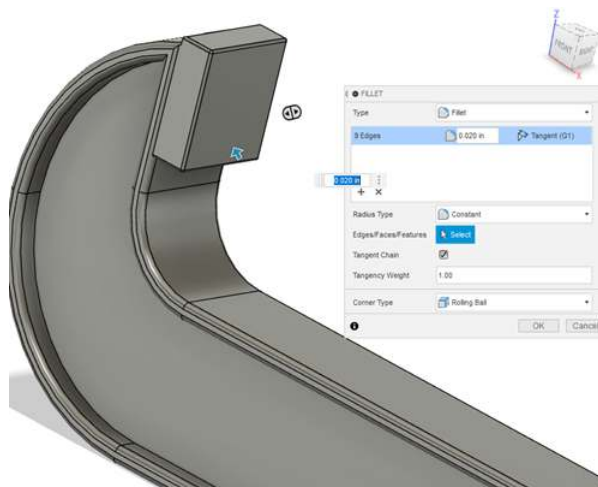


Figure 1.9.11: Fillet applied to edges of the clamping surface.

1.9.2.9 Step 9 - Sketch text on the clamp surface

Select the surface of the clamp and sketch text on the surface as shown in Figure 1.9.12. In the Edit Text dialog box, enter the “TRDS 160” in the first line and “Centralia College” in the second line.

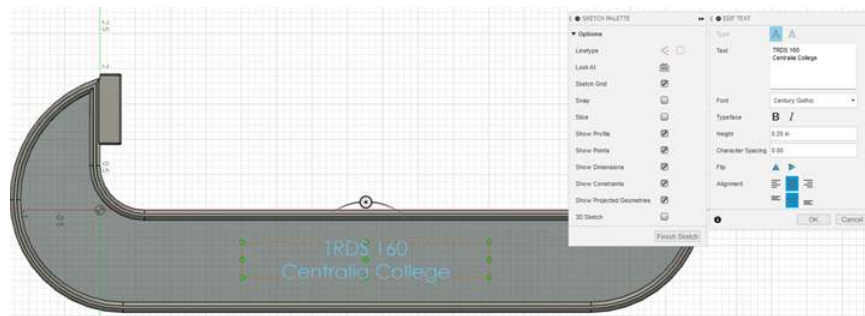


Figure 1.9.12: Sketch text on clamp surface.

1.9.2.10 Step 10 - Extrude text on the clamp surface

Select the sketch text created in the previous step. Extrude the text to a height of 0.20 inches. The extruded text is shown in Figure 1.9.13.



Figure 1.9.13: Extruded text on the clamp surface.

1.9.2.11 Step 11 - Repeat steps 5, 6, and 7 on the other side of the clamp.

1.9.2.12 Step 12 - Sketch a circle on the far side of the clamp.

Sketch a circle of diameter 1.40 inches and with its center at 0.50 inches below the edge as shown in Figure 1.9.14.

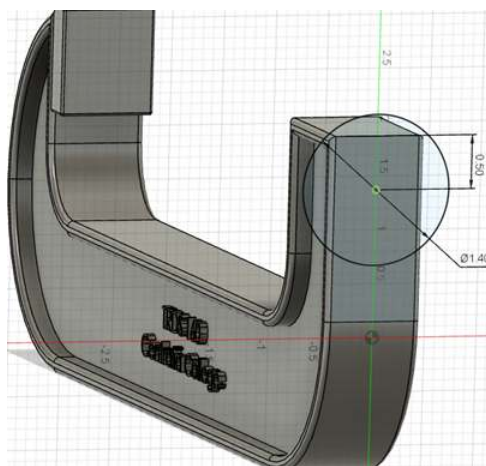


Figure 1.9.14: Circle sketched on the far side of the clamp.

1.9.2.13 Step 13 - Extrude the circle

Extrude the circle created in the previous step to a distance of 1.50 inches as shown in Figure 1.9.15.

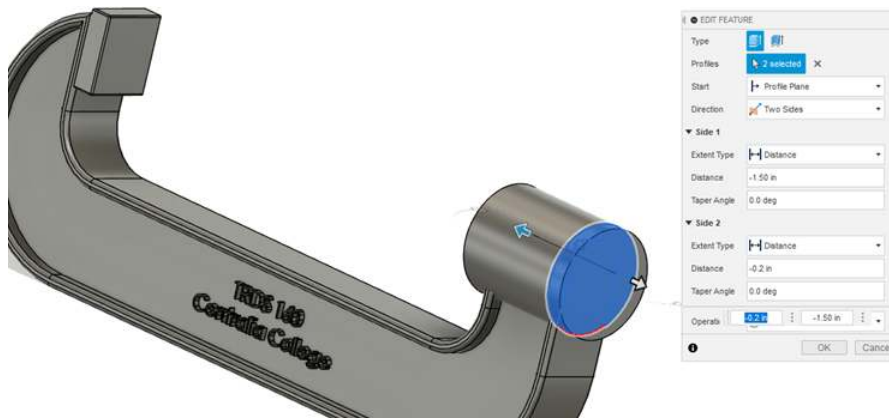


Figure 1.9.15: Extruded circle.

1.9.2.14 Step 14 - Sketch a circle on the feature created in step 13

Sketch a concentric circle of diameter of 1.00 inches as shown in Figure 1.9.16.

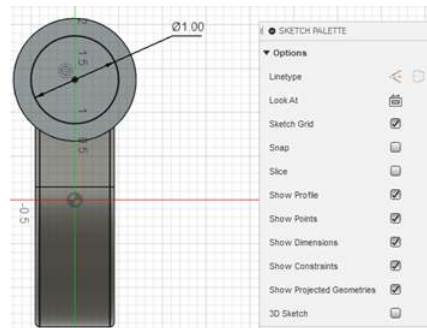


Figure 1.9.16: Sketch of the circle.

1.9.2.15 Step 15 - Extrude a hole from the feature created in step 2.14

Use the Extrude Cut option as shown in Figure 1.9.17 to cut a through hole using the circle created in the previous step.

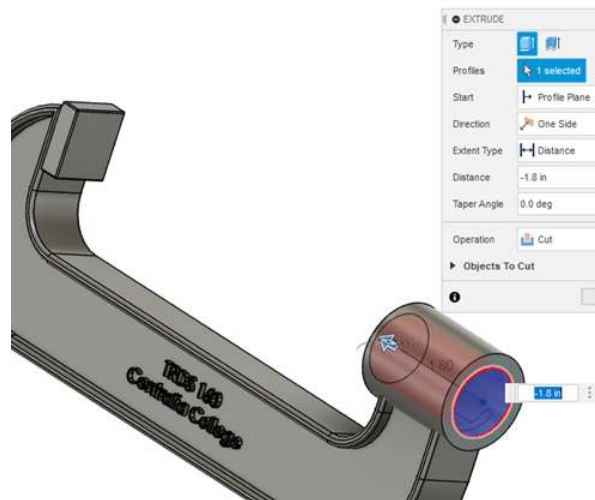


Figure 1.9.17: Extrude Cut option.

1.9.2.16 Step 16 - Create threads on the inside of the hole created in step 15

Add right-handed ACME screw threads of size 1, designation 1, and class 4G, as shown in Figure 1.9.18.



Figure 1.9.18: Add ACME threads.

1.9.2.17 Step 17 - Save this file.

1.9.3 Create the Clamp Spindle

In this section, the clamp spindle will be sketched and then then extruded.

1.9.3.1 Step 1 - Create the sketch of the clamp spindle

Create a new file. Make sure the units are inches. Add a centerline and sketch the profile as shown in the Figure 1.9.19.

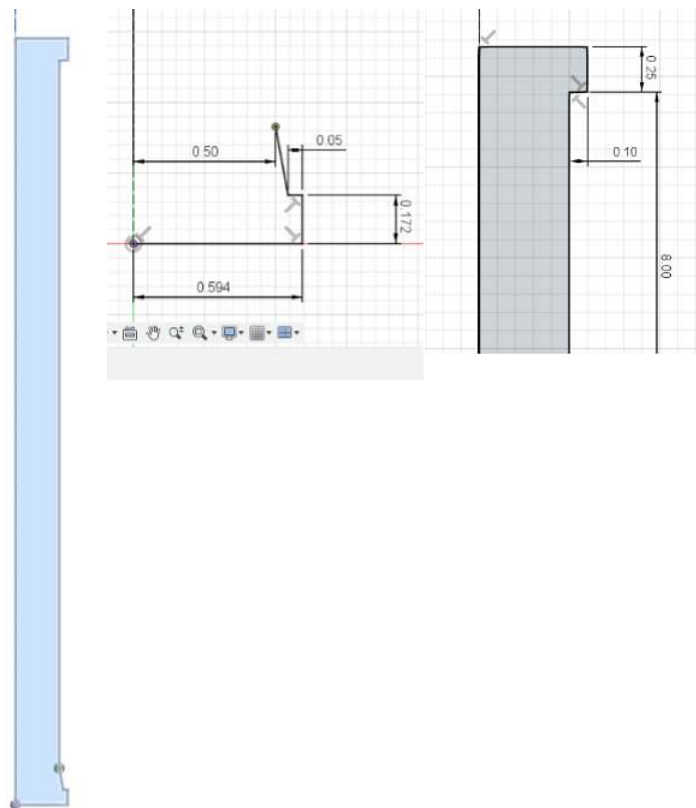


Figure 1.9.19: Profile of the clamp spindle with details of the lower and upper profiles shown.

Step 2 - Revolve the profile created in Spindle step 1

Use the Revolve option to create a cylindrical profile as shown in Figure 1.9.20. To make sure the profile is complete choose an angle of 360°.

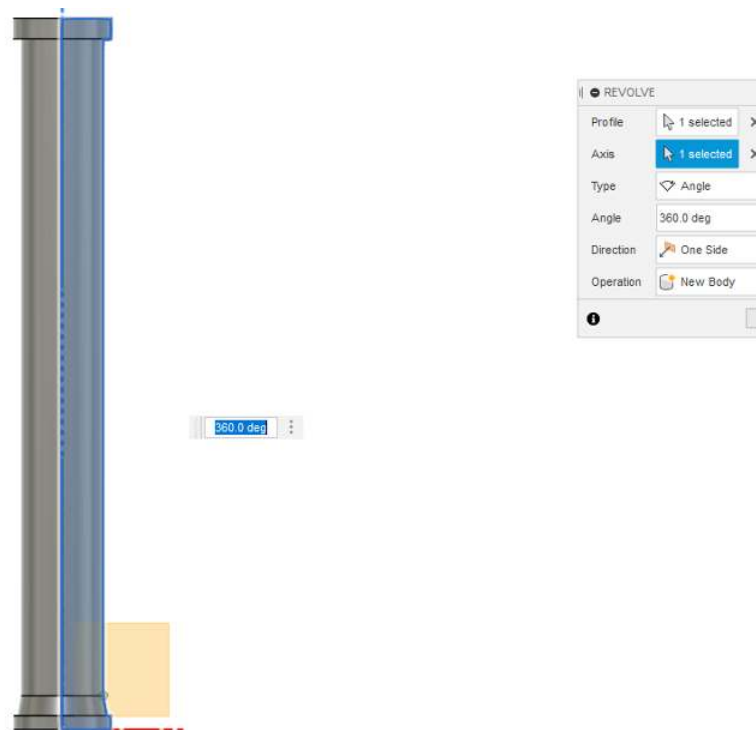


Figure 1.9.20: Revolve to create a cylindrical clamp spindle.

1.9.3.2 Step 3 - Add threads to the clamp spindle

Add right-handed, ACME threads of size 1, designation 1, and class 4G, to the clamp spindle as shown in Figure 1.9.21.

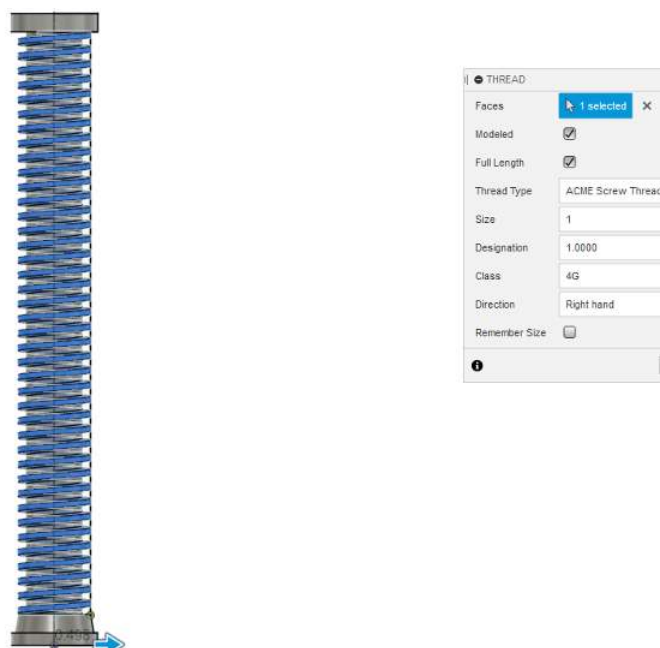


Figure 1.9.21: Add ACME threads to clamp spindle.

1.9.3.3 Step 4 - Sketch a circle on the clamp spindle

Create a sketch of a circle of diameter 0.175 inches that is 0.124 inches away from the top edge of the clamp spindle and 0.60 inches from the side of the clamp spindle as shown in Figure 1.9.22.

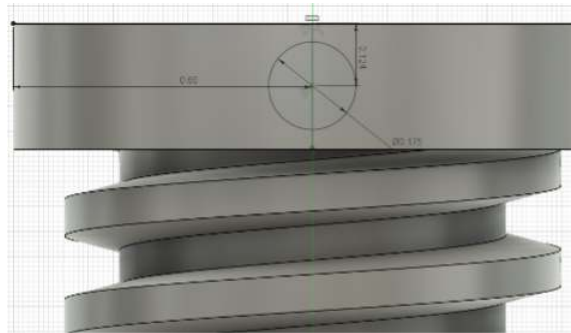


Figure 1.9.22: Sketch the circle on the clamp spindle.

1.9.3.4 Step 5 - Extrude circle to create the swivel bar

Extrude the circle created in the previous step symmetrically to a length of 2.30 inches as shown in Figure 1.9.23.

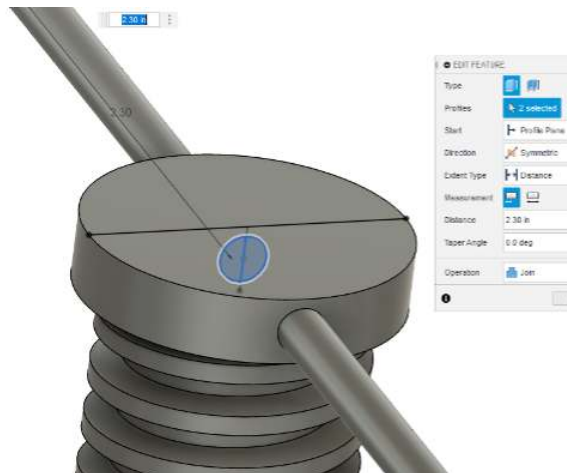


Figure 1.9.23: Extrude the circle to create the swivel bar.

1.9.3.5 Step 6 - Create fillet at the edge of the swivel bar

Create a fillet of radius 0.050 inches as shown in Figure 1.9.24.

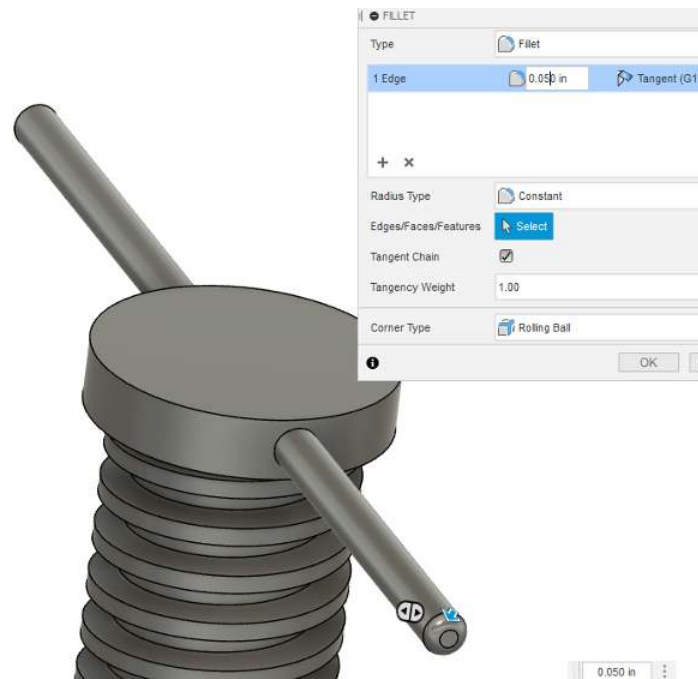


Figure 1.9.24: Fillet at the end of the swivel bar

1.9.3.6 Step 7 - Save this file.

1.9.4 Assemble the body clamp and clamp spindle

In this section the body of the clamp and clamp spindle will be assembled as shown in Figure 1.9.25.



Figure 1.9.25: Assembled clamp body and clamp spindle.

1.9.4.1 Step 1 - Insert components in the new file

Create a new file and add the clamp and the spindle by using the Assemble option as shown in Figure 1.9.26.



Figure 1.9.26: Assemble option to add components into the file.

1.9.4.2 Step 2 - Create Joint

Use the Joint command and select the origin on the top side of the spindle and the origin located on the end of the spindle hole as shown in Figure 1.9.27.

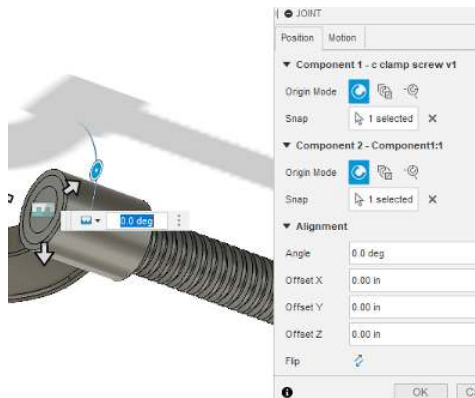


Figure 1.9.27: Creating the joint.

1.9.4.3 Step 3 - Edit Joint

Right click the joint to edit the parameters. Select a cylindrical joint and edit the rotational and linear joint limits as shown in Figure 1.9.28.

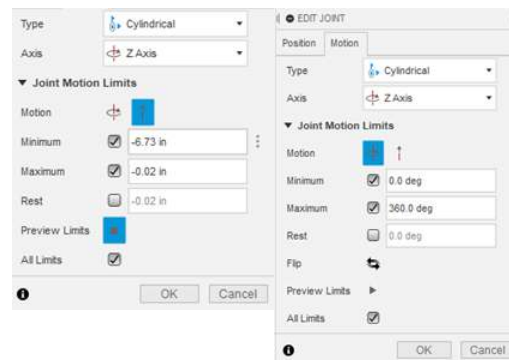


Figure 1.9.28: Editing the linear and rotational joint.

1.9.4.4 Step 4 - Verify the joint parameters

Right click on the joint and select the Animate Joint option as shown in Figure 1.9.29. This will animate the joint and verify if the parameters worked out as required.

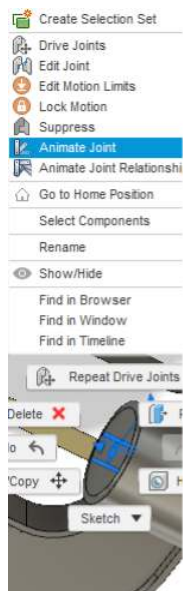


Figure 1.9.29: Animate Joint option to verify the joint parameters.

1.9.4.5 Step 5 - Save the file.

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