MEC-07: Computer Design Fundamentals

Student Name

Table Of Contents

ABOUT THIS WORKBOOK

OVERVIEW

The MEC-07: Computer Design Fundamentals workbook is designed to supplement the Mechanical Design Curriculum at Exceed Robotics. This coursebook provides reference materials, examples, and homework problems for extra practice. Students using this workbook are encouraged to follow and reference all the course material. Along with completing the homework problems to enhance their understanding of Mechanical Design and CAD (Computer Aided Design).

Each class has material that contains:

1. Content Explanations
2. Class exercises to follow
3. Homework

The summary teaches students the topics covered in weekly classes. Class exercises give students something to follow and practice their learning. Homework is extra problems and practice to check the students understanding of the material. Watching the videos and checking off homework as you go are highly encouraged!

USING THIS WORKBOOK

This workbook has features with which you can interact. Using the Table of Contents, you can jump to the parts that you want to start at. Along with how the homework sections are editable. When making edits to answer questions, do not forget to save your work! To save your workbook at any time, you can use **Ctrl+S** for windows or **Cmd+S** for MacOS. Although when you are exiting, you will receive a prompt to save if you have not. When there is link like: CLICK HERE, use **CTRL+Click** to open it.

COURSE INTRODUCTION

How are cars, computers, medical devices, or machines designed?

Engineers and Designers must first conduct research and ask what requirements are required for the project. After conducting research, they start to develop ideas for their project.

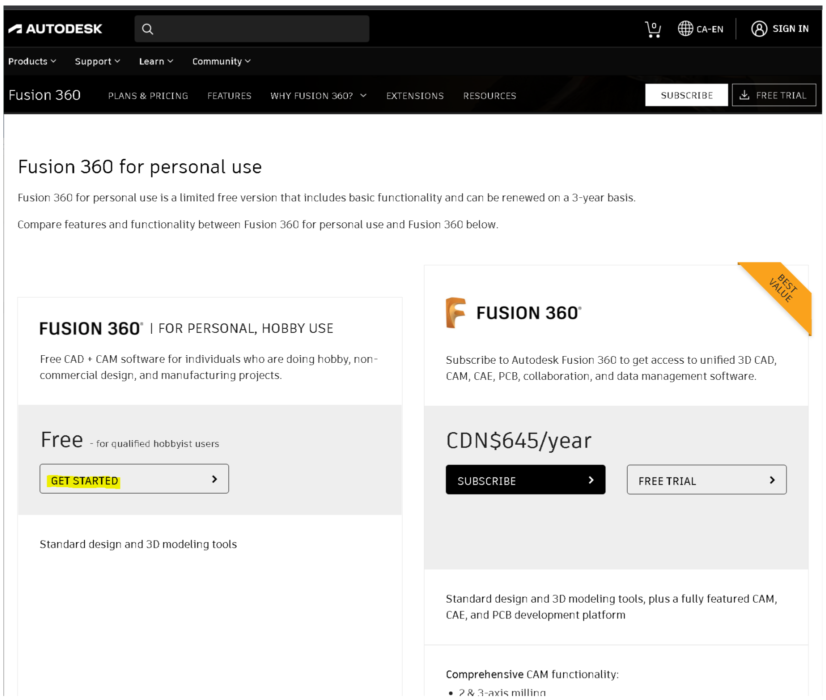
Once they are ready to design, they use a computer aided design (CAD) software to model parts for their project. The CAD software you will be using this term is named Fusion360.

SETTING UP THE ENVIRONMENT

Autodesk Account and Fusion360 Installation **IMPORTANT**

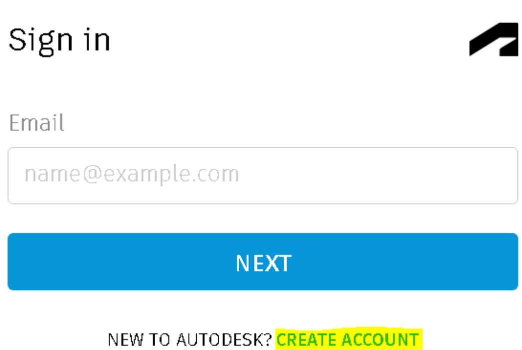
Please create an Autodesk account following these steps:

1. To start CLICK HERE
2. Click on "**Get Started**" under FOR PERSONAL, HOBBY USE



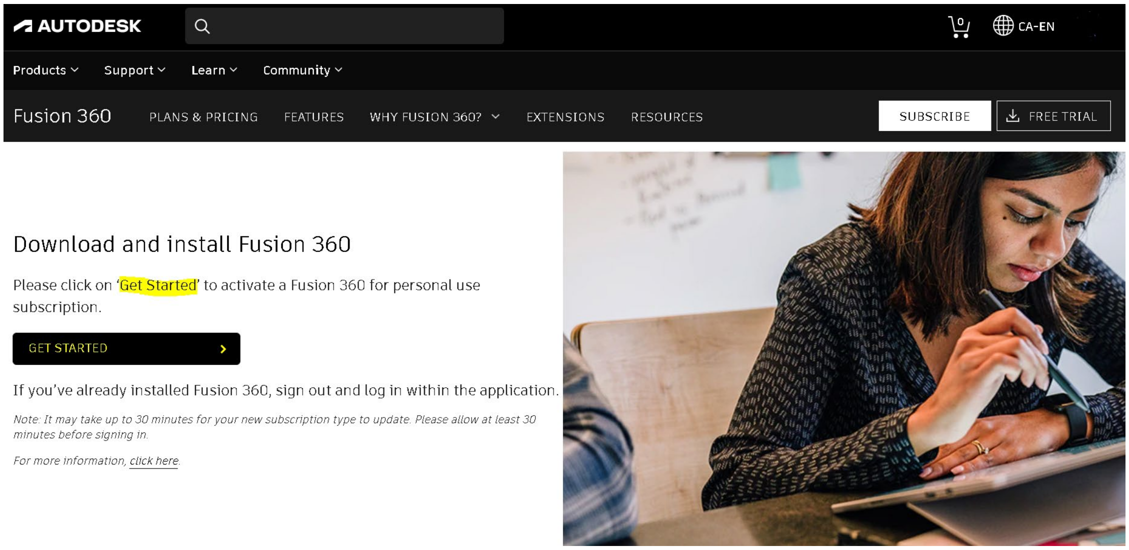
3. Click "**Create Account**" and complete registration.

You can ask your parents for assistance when creating an account.

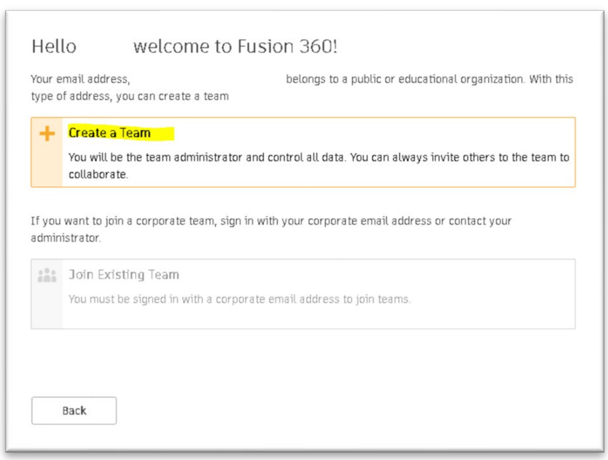


Be sure to fill out all the information!

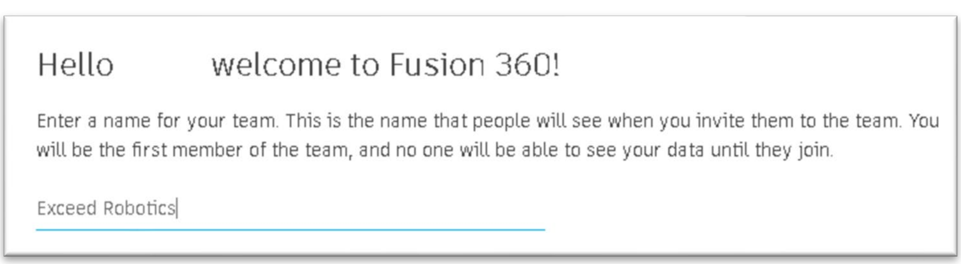
4. Click Download Fusion360 and follow the steps



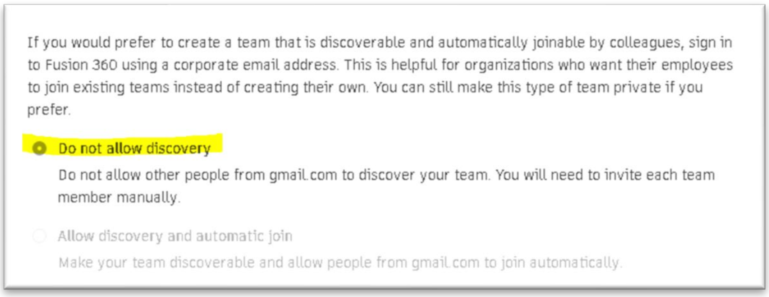
5. When starting Fusion360, select create a New Team and DO NOT make public/available.
    a.



    b.



c.



The press "**Create**" and begin!

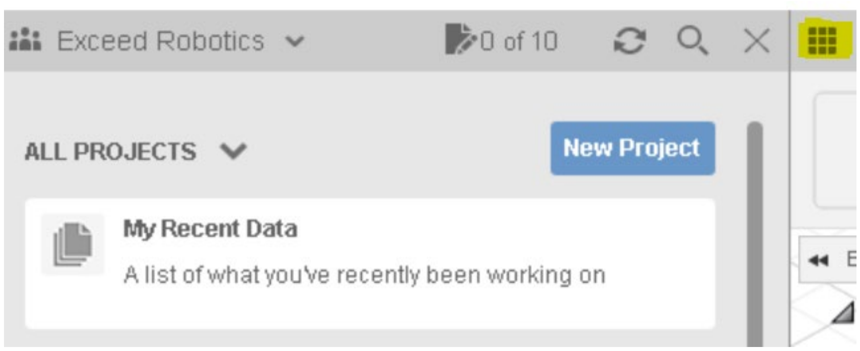
Congratulations! You have just started your journey in MEC-07.

BEFORE STARTING

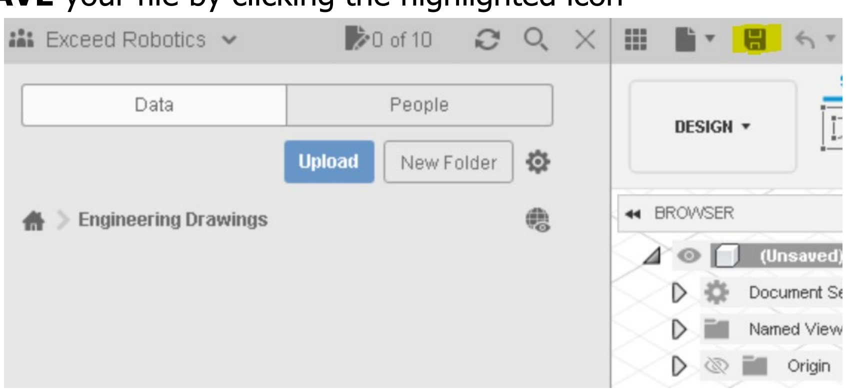
FILE ORGANIZATION

CREATING A NEW PROJECT:

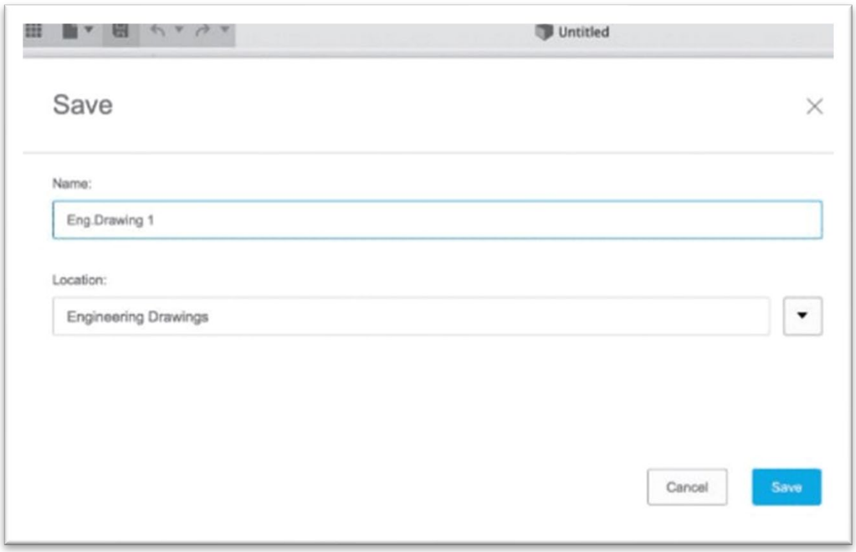
1. Click the **DATA PANEL** (highlighted object) and then click "**New Project**"



2. Create a new project called "**Engineering Drawings**"
3. Next **SAVE** your file by clicking the highlighted icon



4. Call it "**Eng. Drawing 1**"

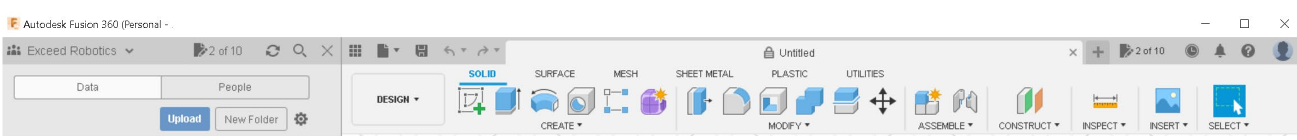


Click "**Save**" and you are now done!

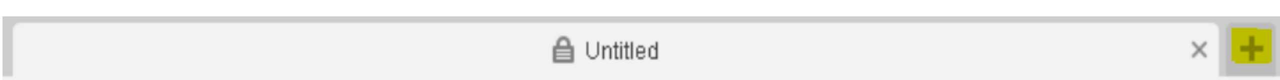
This is the process of <u>creating new project folders</u> and <u>saving your creations</u>. You can use this to organize new projects you create. Along with saving and naming your creations to stay organized. **You can always refer to this section to know how to organize your Fusion 360 workspace**.

CREATING A NEW DESIGN:

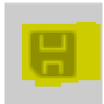
1. At the top of your Autodesk Fusion 360 application, all your designs are shown at the top

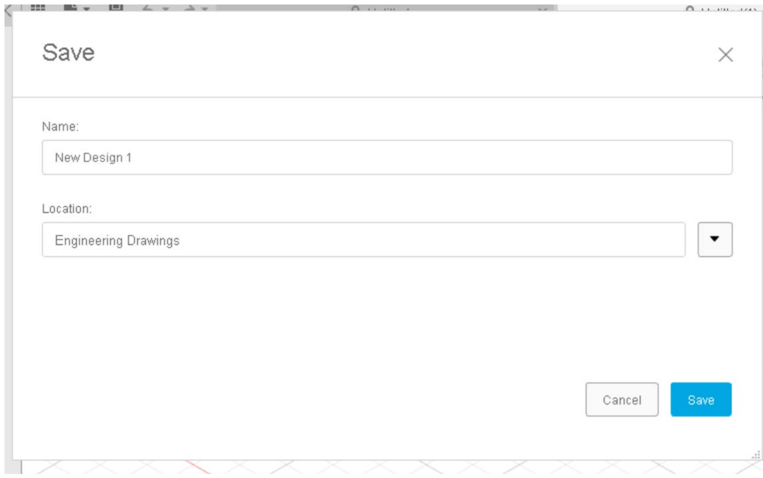


2. Press the "**+**" button to make a new design



3. **Save and name** your new design file

Press this button:  and then **name and save** your file:

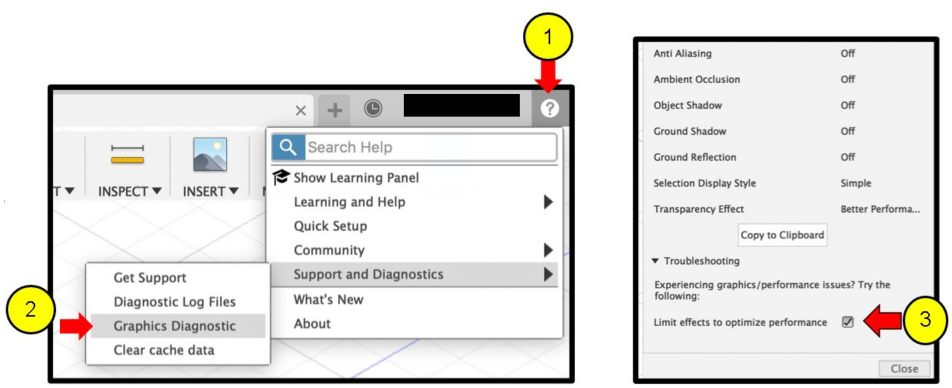


4. It will now be saved and ready to use!

OPENING DESIGNS:

To open any design you create, just double click the project when you see it on the left!

OPTIMIZING PREFORMANCE:



ENGINEERING DRAWINGS

Engineering drawings are technical drawings that have all the information needed to produce a product or system.

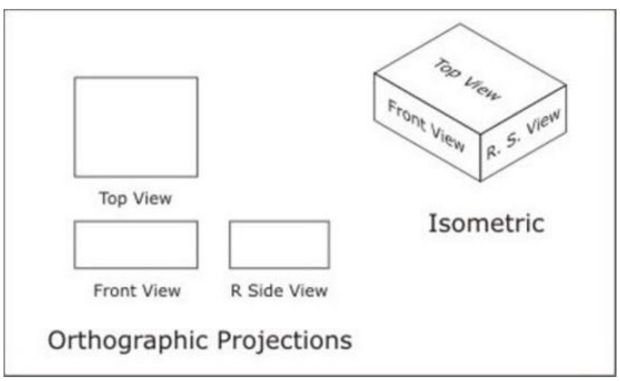
Drawings will contain dimensions, material required, and any necessary notes needed.

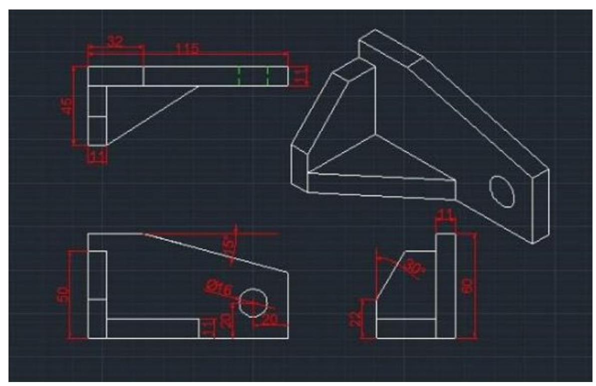
For the next three classes, we will be covering how to read drawings and design parts from reading drawings.

TYPES OF VIEWS

ORTHOGRAPHIC DRAWINGS

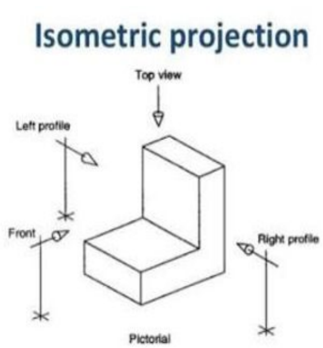
- An orthographic view takes a 3D view and displays it as 2D views. This type of drawing allows you to see details of the part from the front, side and top view.

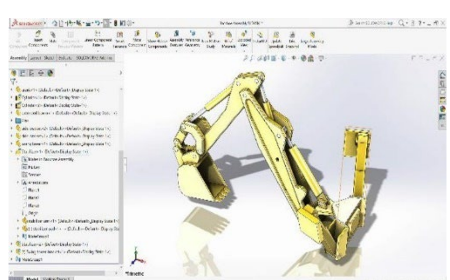


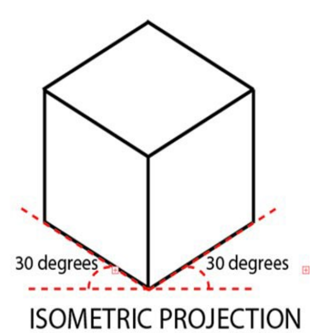


ISOMETRIC PROJECTION

- An isometric projection is a view of an object at a common 30-degree angle. This view is commonly used to view multiple sides of an object.

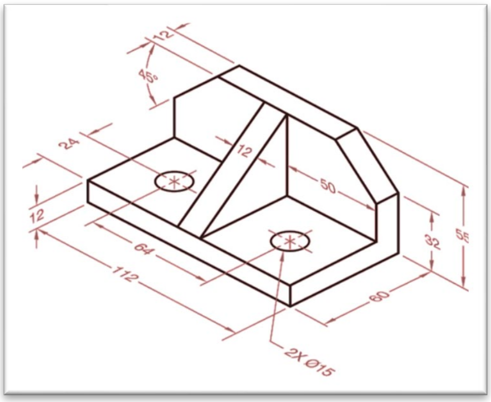






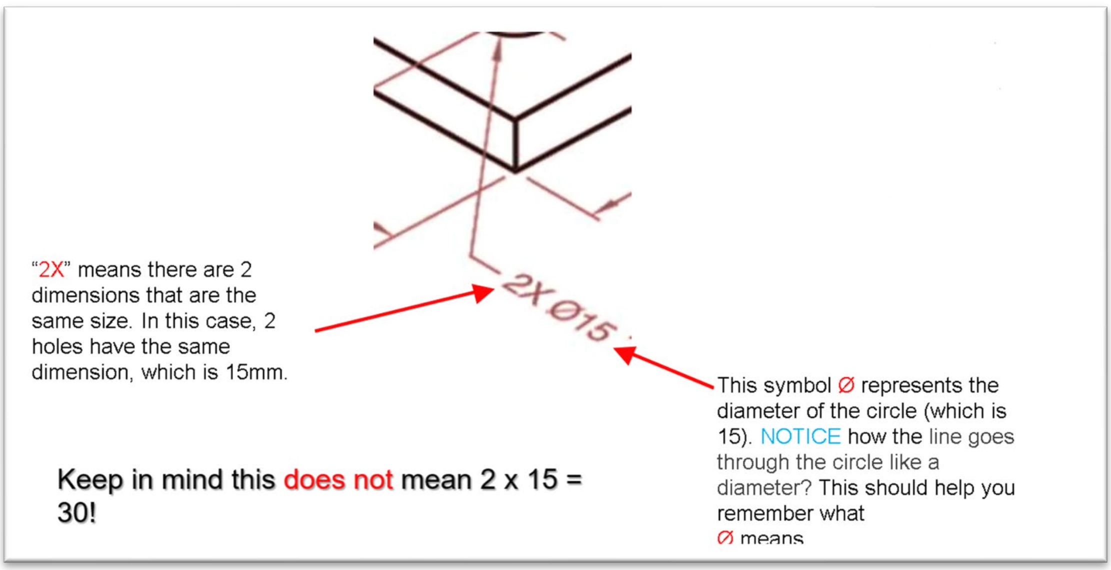
READING DESIGNS

Look at this drawing:

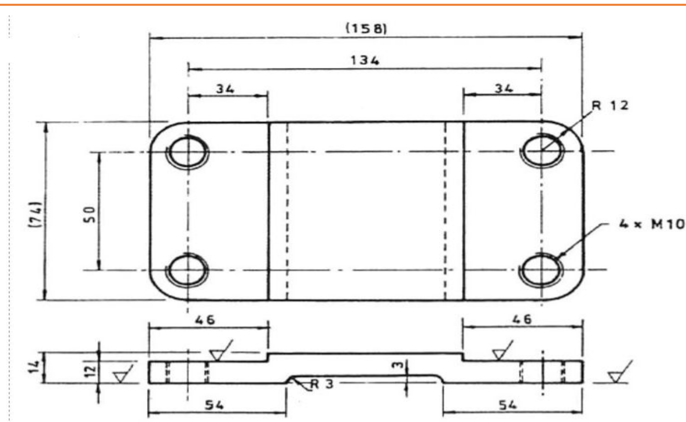


What is the diameter of the holes?

Answer: 15mm



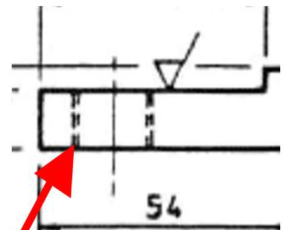
Look at this drawing:



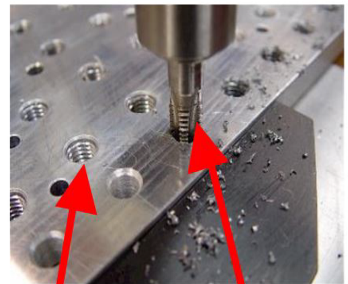
What is the dashed lines in the holes?

Answer: Represents how the whole width is cut straight through. Creating a full hole.

Answer: All the way through the part (this is called a "through all hole")

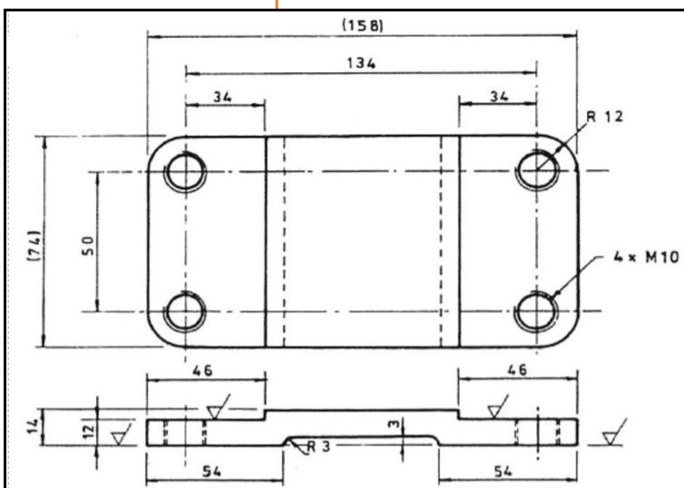


These dashed lines means the holes are **threaded** and since the dashed lines go to the bottom of the part, it means the holes go all the way through.



This is an example of a hole that has threads on the inside. This will allow for screws to sit in the hole.

This is called a thread tap. This tool can create threads in a hole. This can be done manually by hand, or by a machine.

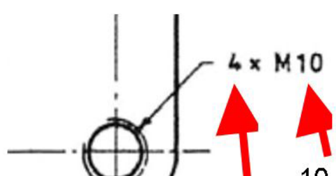


Looking back at the drawing

What is the diameter of the holes?

Answer: 10mm

Answer: 10 mm.

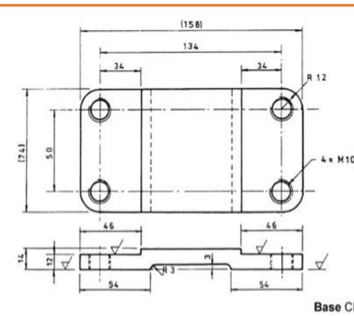


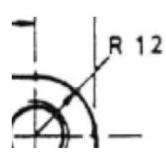
10 mm diameter

4 holes

M10 represents a screw notation (M stands for metric). The 10 is the diameter for the hole.

The 4 means there are 4 holes that have the same dimension.





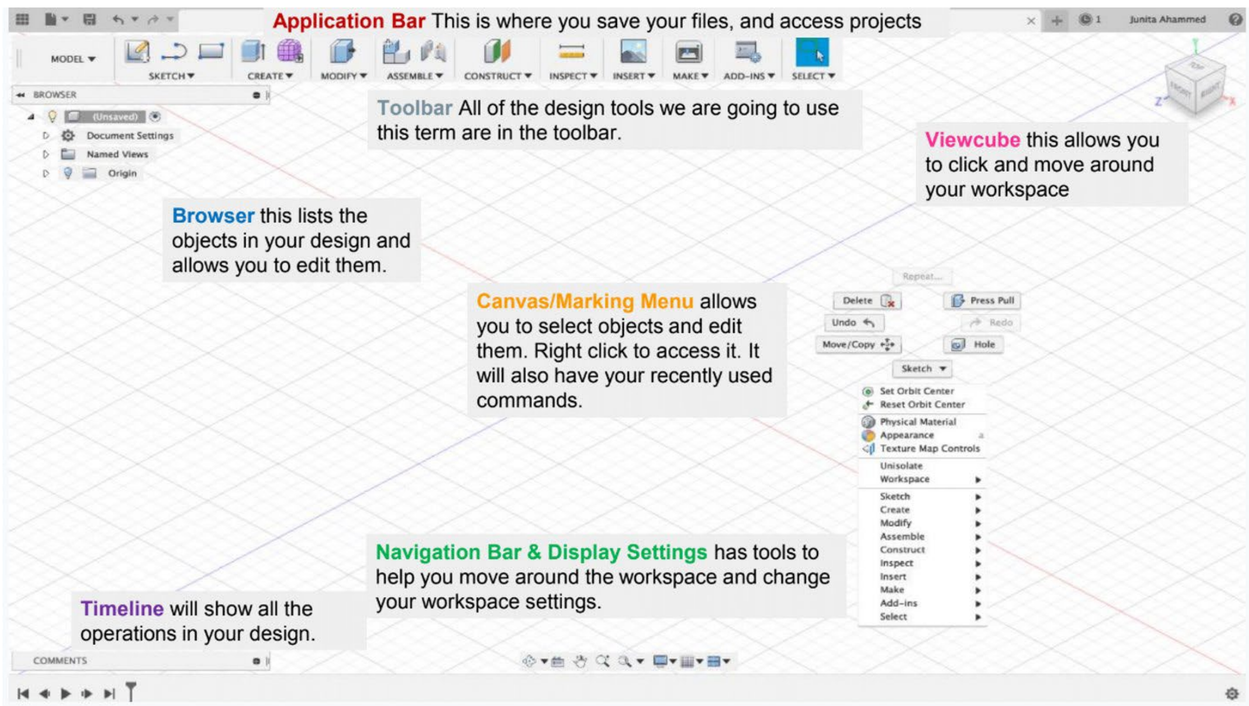
The "R12" is referring to the rounded corners on the part. We will be using the fillet tool to create those rounded corners.

1: INTRODUCTION

FUSION 360 ENVIRONMENT

Using the file, you made here, explore, and familiarize yourself with the tools and areas listed down below.

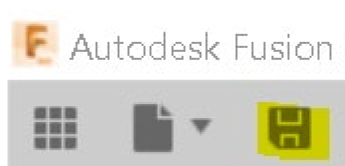
These are common tools/areas that you will be using throughout the course.



ICONS TO REMEMBER:

If you ever want to see your files, click the **Data Panel** at the top left:

If you ever want to save your work, click the **save or Ctrl-S** at the top left:



IMPORTING FILES

CLASS VIDEO:

Watch this video and complete the exercise(s)

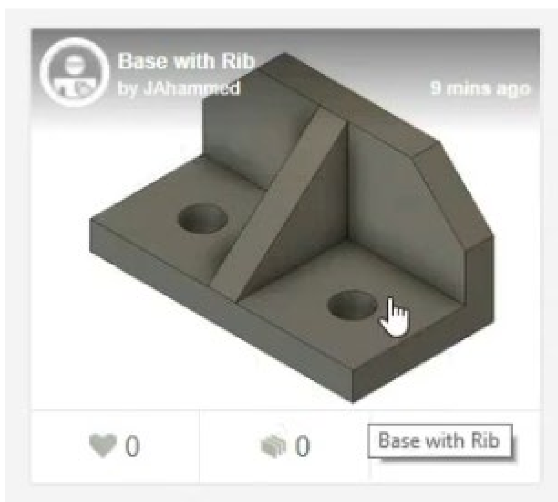
CLICK HERE

Otherwise follow these steps:

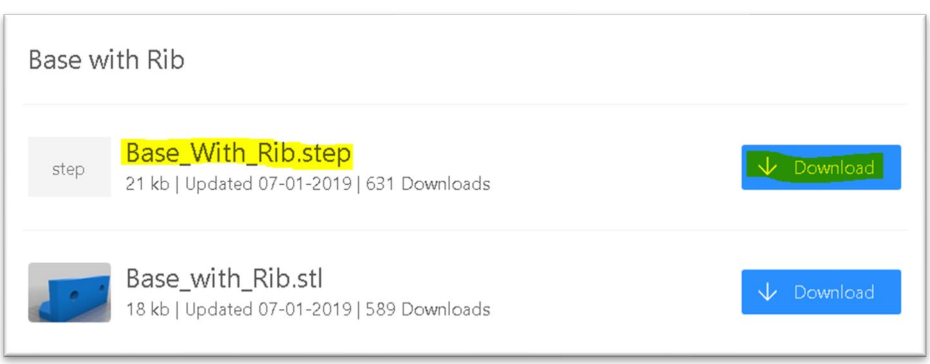
STEPS TO FOLLOW:

1. Go on google and search up "**thingiverse.com**"

2. Once you are on "Thingiverse", use their search bar and type "**Base with Rib**"
3. **Select** on the object that looks like this:

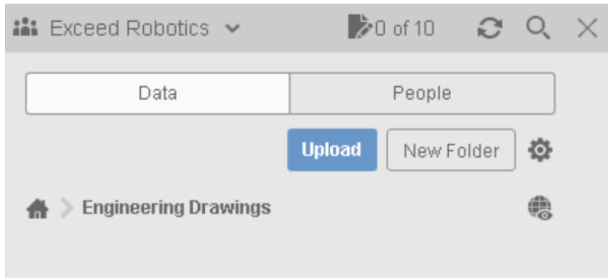


4. Scroll to the bottom of the page and click "**Download**" on the .step file



5. Drag the file onto your desktop (You can delete it at the end of these instructions)

6. Open Fusion and click "**Upload**" near the top left.



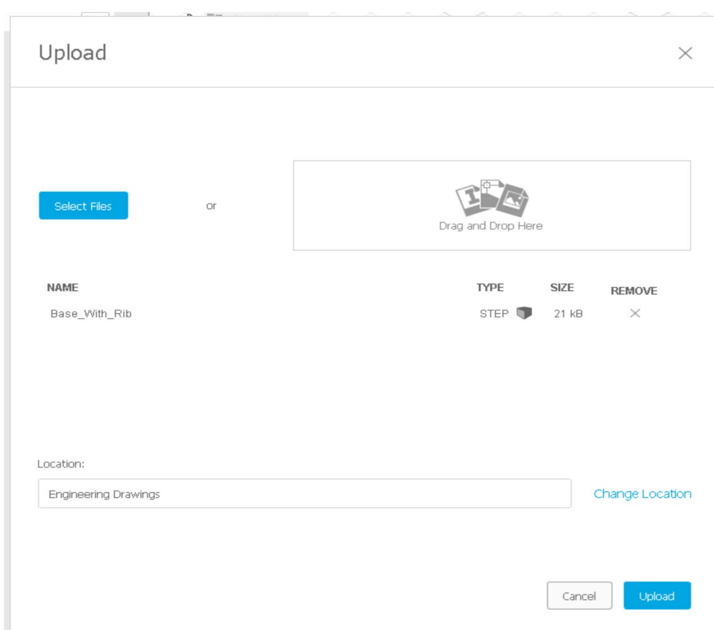
7. Click "**Select Files**"

8. Files will open, now click "**Desktop**," and find the **.step file** you downloaded

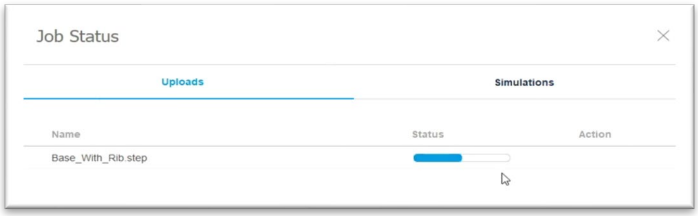
This is the file you are looking for:

9. It will now show you which file you have chosen, double check it is the:

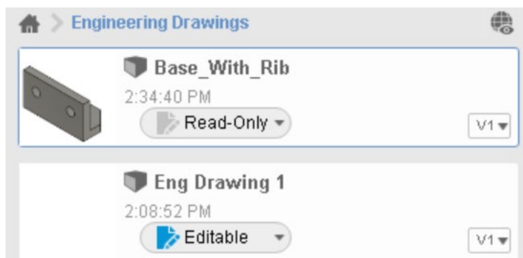
Base_With_Rib STEP



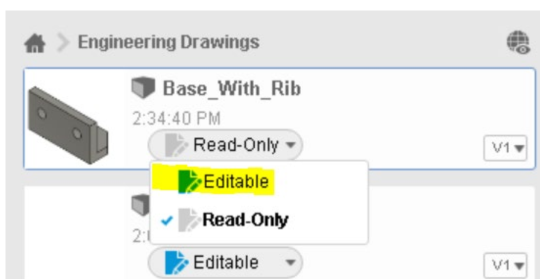
10. Once you are sure, click "**Upload.**" This is what you will see after you click Upload.



11. When it has finished uploading it will say "**Complete**" and show up on the left side



12. Click on the **Base_With_Rib** that you now see on the side, click "**Read-Only**" and then make it editable by clicking on "**Editable**"



13. Now save your workspace by clicking this icon:  or clicking **Ctrl+S**

You can now <u>click on the file (**Base_With_Rib**) to open it</u> and see the part you have uploaded! You can click on the part so you can see dimensions, rotate it, and truly mess around with all the tools at the top.

REFER BACK TO [FUSION 360 ENVIRONMENT].

After messing around, if you **did not save** any changes you made, **you can go back to the original part by closing and reopening Autodesk Fusion 360**. Otherwise, you can restart this process to upload the part again!

NAVIGATION TOOLS

CLASS VIDEO:

Watch this video and complete the exercise(s)

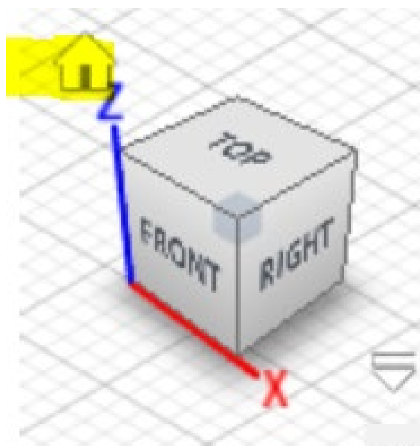
CLICK HERE

Otherwise follow these steps:

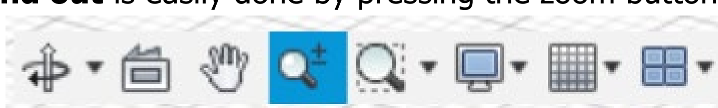
STEPS TO FOLLOW:

In these steps you will see the different tools you can use in Autodesk Fusion. When following these steps **try it out on the part you uploaded above**!

1. If you ever feel confused onto where you are looking, Click the "**Home Button**" to go back to the original position. This area is also called the ViewCube



2. **Zooming in and out** is easily done by pressing the zoom button



Once you have selected the zoom button, **left click and drag your cursor around to either zoom in or out**. Otherwise, you can use the scroll wheel.

SCROLL 	**Scroll** the middle mouse wheel to zoom in or zoom out.

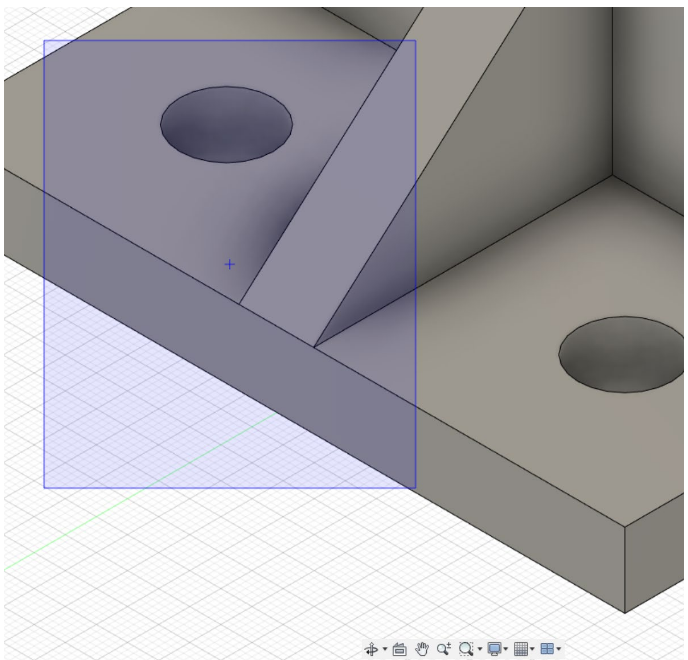
3. **Fitting the object** to your project space can be done by pressing **F6**

4. The "**Zoom Window**" allows you to select an area you want to zoom into.

When using the zoom window, you can left click and select highlight a part you want to zoom into. You can do this by:

<u>Clicking the Zoom Window</u>:

Select an area by left clicking and dragging your mouse to show this rectangle. Prompt the rectangle over the space you want to zoom in and it will zoom in automatically once you let go of your mouse.



The image above shows how the rectangle should look when selecting and area

5. Click the "**Orbit Tool"** and use it to move around

By Left-Clicking and dragging your mouse either up, down, left, or right, you can move your point of view around to see every angle of your part.

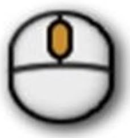
You can also use **SHIFT+SCROLL WHEEL** to orbit the view

SHIFT KEY + 	Hold the **SHIFT** key and **click and hold** middle mouse button to orbit the view.

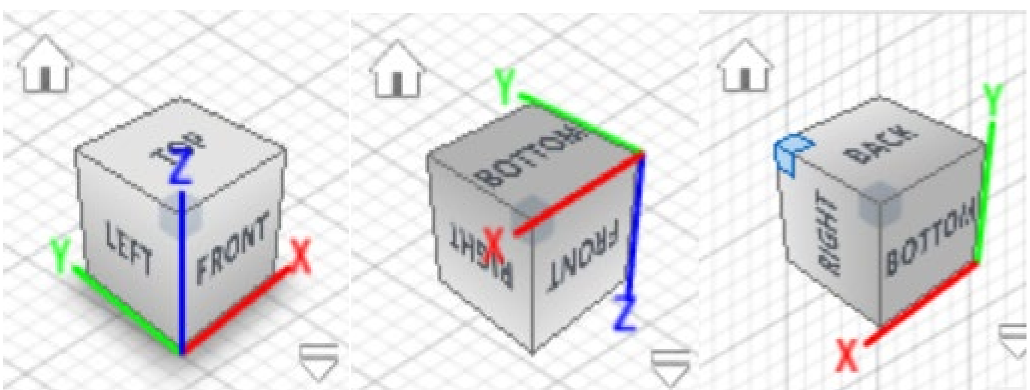
6. Click the "**Pan Tool**" and use it to move around

By Left-Clicking and dragging your mouse either up, down, left, or right, you can move your view of the part without changing the angle of your view.

You can also do this by holding your scroll wheel and dragging your mouse around.

HOLD 	**Click and hold** the middle mouse button to pan the view.

7. You can **view different sides** of your part by click sides of the view cube.



You can click <u>Front, Back, Top, Bottom, Right, Left, and The Corners </u>to change the point of view of your shape. If you want to go back to the original position, use the Home Button again!

SKETCHING AND EXTRUDING

CLASS VIDEO:

Watch this video and complete the exercise(s)

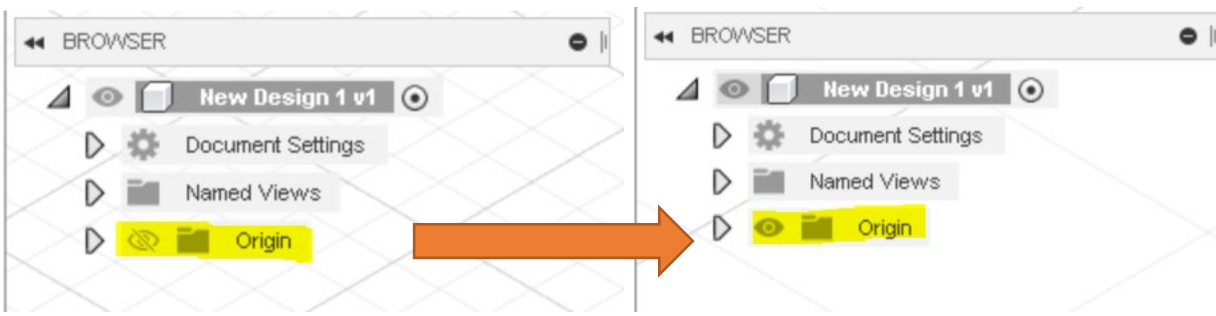
CLICK HERE

Otherwise follow the content below:

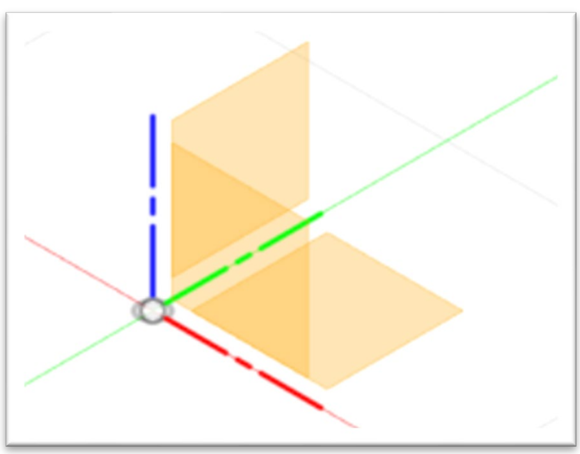
GETTING STARTED

1. Create a new project following the steps in File Organization: Refer to CLICK HERE

To Sketch in Autodesk Fusion 360, you must select a plane to draw on. To see these planes, make sure your "**Origin**" has its **eye open.**



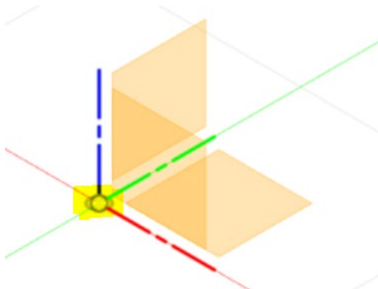
You will now see:



The Sketch Planes!

The Sketch planes help us choose which side or direction our part will be created in.

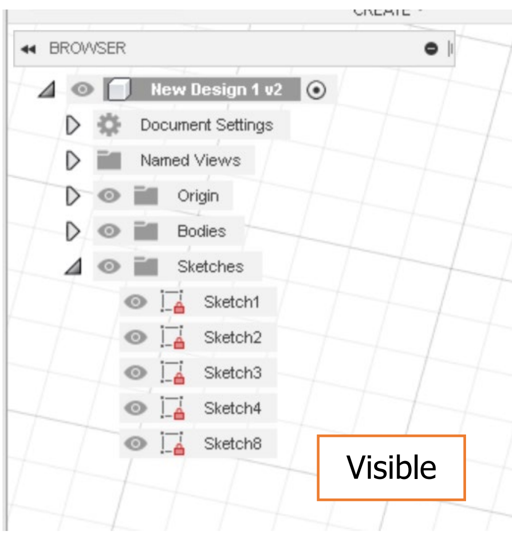
When sketching it is good practice to start your sketch at the center point/origin (0,0).

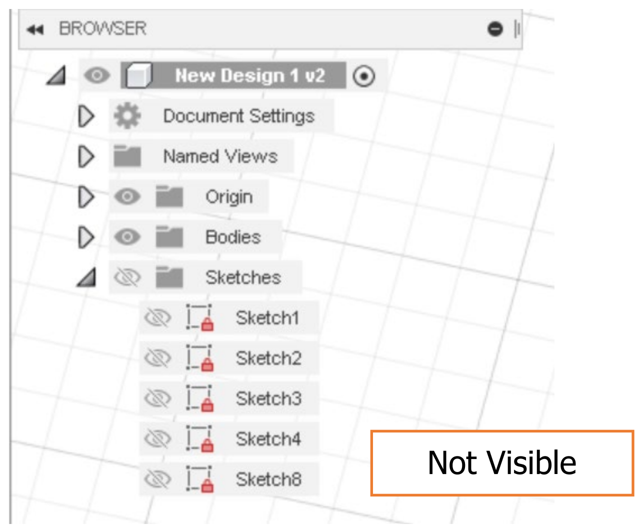


The origin is highlighted in this photo.

KEY THING TO REMEMBER:

Keep your sketches VISIBLE (eye open)





USEFUL COMMANDS:

Undo

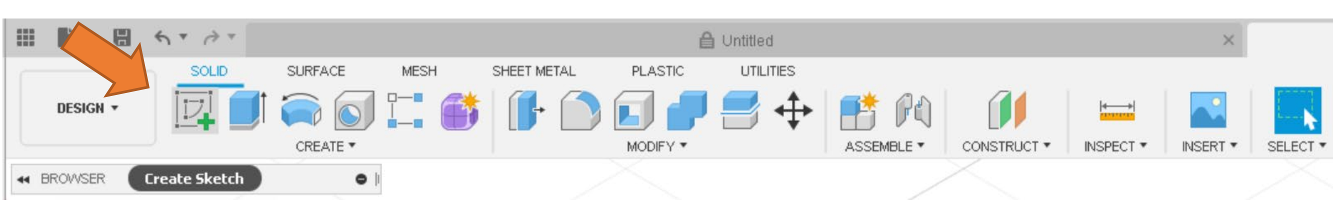
CTRL+Z - to undo your previous action, press Ctrl followed by Z

Redo

CTRL+Y - to reverse your undo, press Ctrl followed by Y

SKETCHING:

To sketch <u>press the plane you want</u> and then click "**Create Sketch**"



If you have correctly done this, the top of your program will now look like this:

Above "**Create**" those are your sketch tools, you can free draw, make rectangles, circles, triangles and use a symmetrical tool to copy sides.

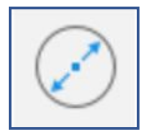
Please COMPLETE ALL 3 EXERCISES!

EXERCISE 1: Making a circle with the diameter of 10cm

Make a New Design and name it!

Now for this example, click the circular sketch icon

and start drawing from the origin.



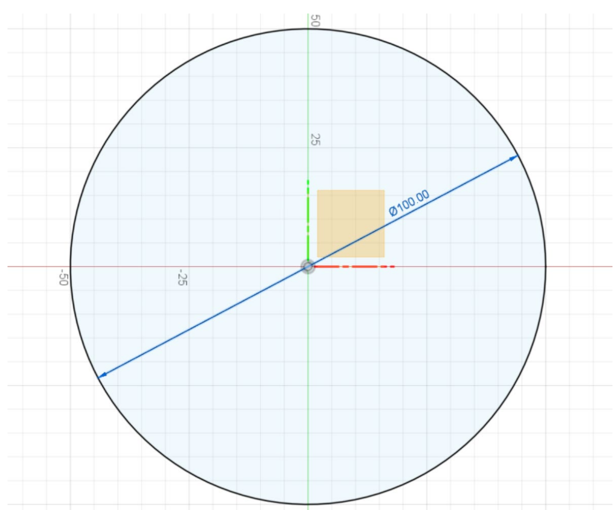
<table>
<tr>
<td>
Origin:</td>
<td>
1. Start by clicking the origin point

2. Drag out your cursor to the desired size (10cm)
OR
3. Type in the dimension you want, (10cm)

Be aware of the measurement size,
Autodesk fusion shows the measurement and size in millimeters (mm)

Hint: 1cm = 10mm
</td>
</tr>
</table>

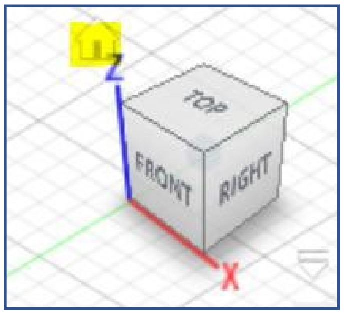
The Final product should look like:



Once you have made this sketch, click "**Finish Sketch**", and let us now extrude!

Save your work
by hitting the
save icon!

EXTRUDING:

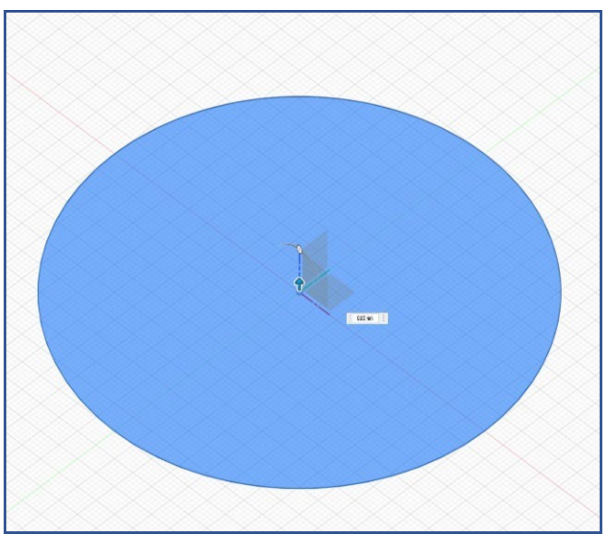


When out of sketch mode, **always** press the Home Button in the View Cube to properly view the 2D sketch.

This is good practice!

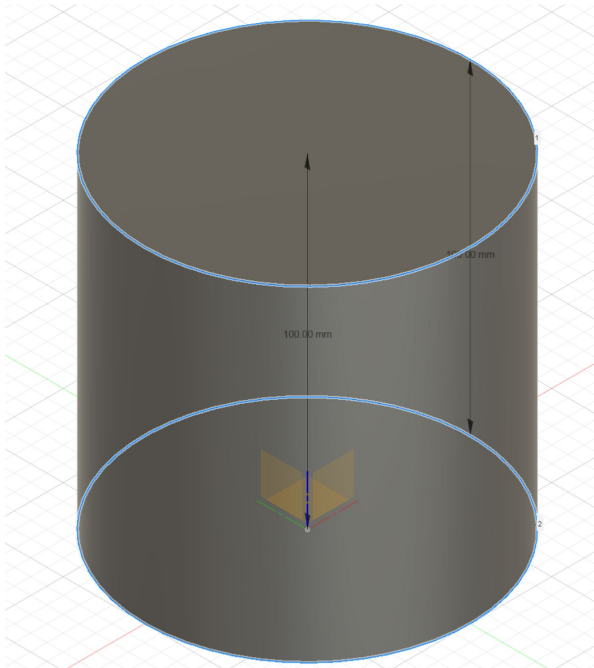
Now that you can see your sketch, press the "**Extrude**" icon**,** and click your sketch

You should now see this:



(The sketch is highlighted and has a little arrow in the middle)

To now make the 2D shape into a 3D shape **bring the arrow up or type the size you want**. In this example, the circle is extruded to 100mm.



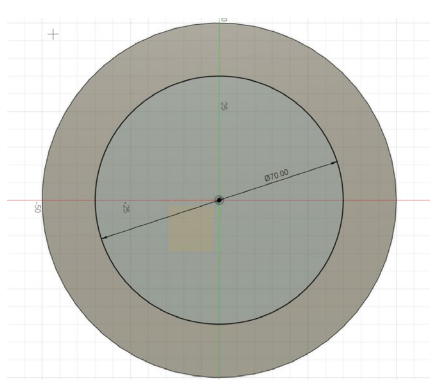
Congratulations, you have now extruded your shape!

GOING FURTHER:

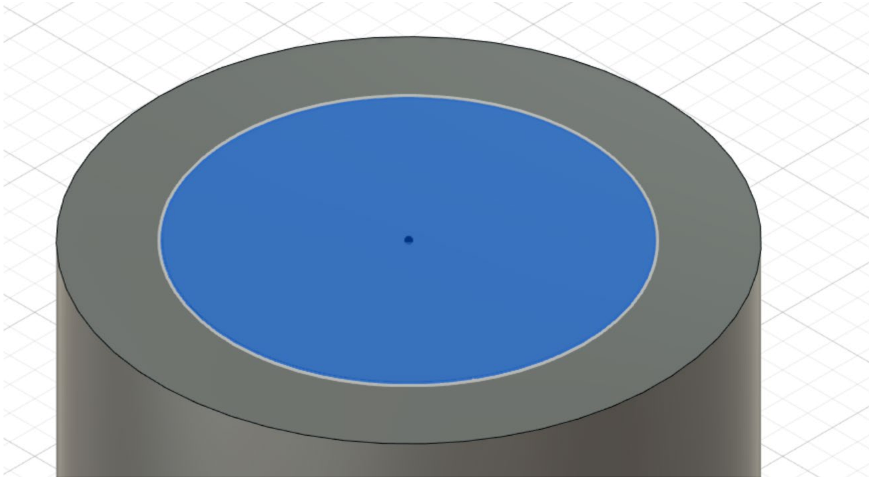
Now we are going to make a cut in this cylinder to give it a hole.

To do this:

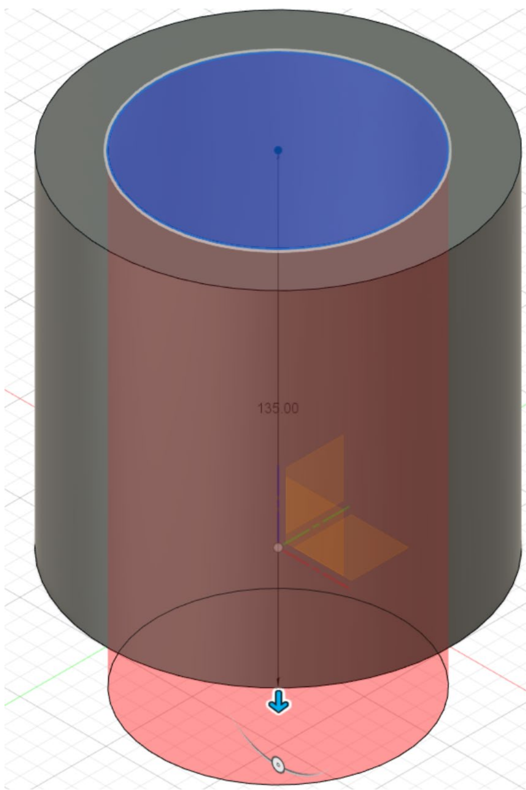
1. Click the <u>top face of your cylinder</u>
2. Click the "**Create Sketch**" button at the top
3. Click the "**Center diameter circle**" button above create
4. Draw a circle from the origin with a smaller diameter than the cylinder. (If you have followed this example, make the sketch have a diameter of 70mm)



5. Press "**Finish Sketch"**
6. Click the circle you have drawn at the top (One portion should be highlighted blue)



7. Click the "**Extrude**" button
8. Drag the arrow downwards, the **area should turn red when MAKING A CUT**

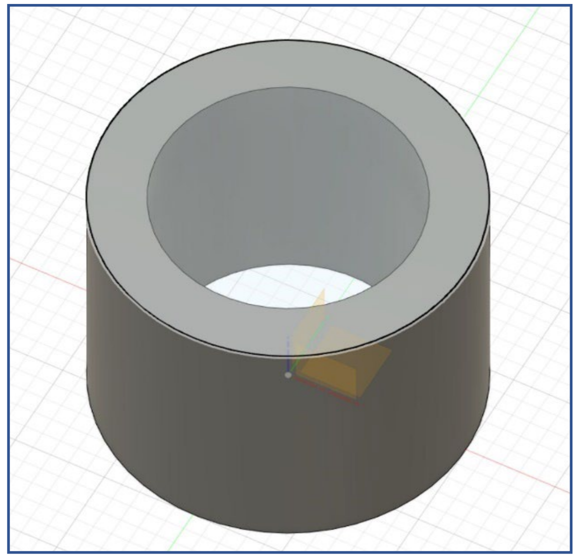


You can make the cut as deep as you would like.

9. Once you have made the cut, press enter, and the cut will be finalized!

You can now see how the part has a hole.

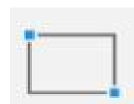
Congratulations! You have now used the main functions of **Extrude**

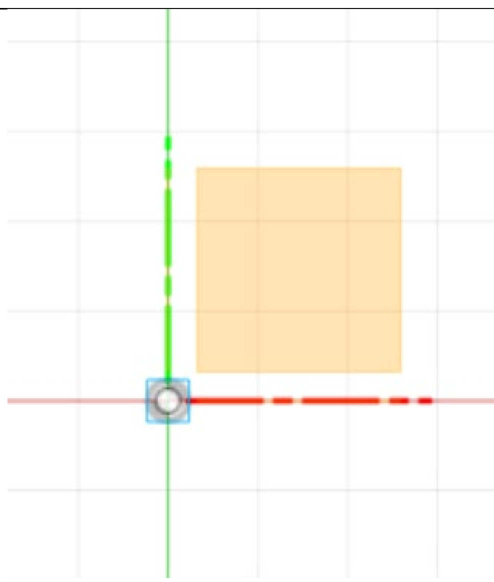


Save your work by hitting the save icon!

EXERCISE 2: Making a rectangle (10cm by 15 cm)

For this example, click the plane you want to draw on and click the "**Create Sketch**" button. Make a New Design and name it!

Start from the origin and click the "**2-point Rectangle**" button 



Origin:

1. Start by **clicking the origin point**
2. Drag out your cursor to the desired size (10cm x 15cm)

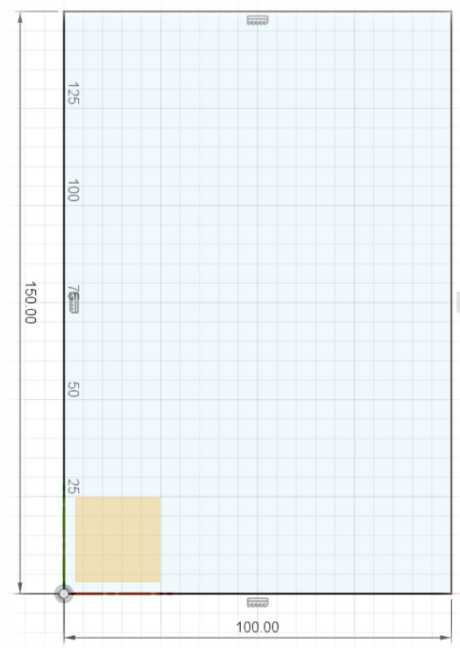
OR

3. Type in the dimension you want, (10cm or 15cm) and then press tab and type in the next measurement

Be aware of the measurement size, **Autodesk fusion shows the measurement and size in millimeters (mm)**

Hint: 1cm = 10mm

The Final product should look like:

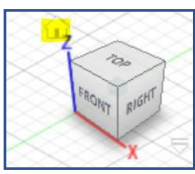


Once you have made this sketch, click Finish Sketch.

Now save your work by hitting the **Save Icon!**

<u>Now we can extrude!</u>

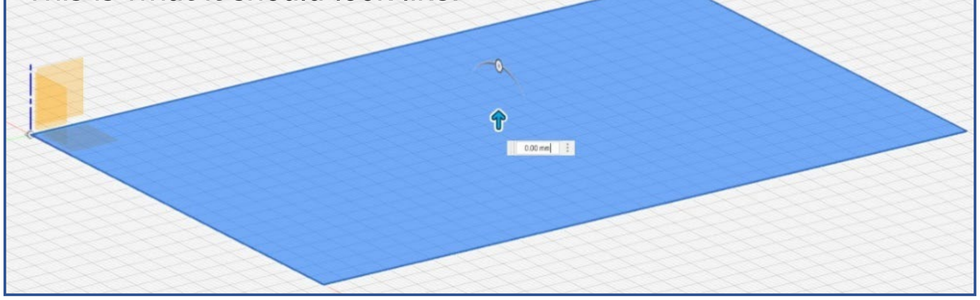
EXTRUDING:

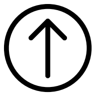


When out of sketch mode, <u>**always**</u> press the Home Button in the View Cube to properly view the 2D sketch. This is good practice.

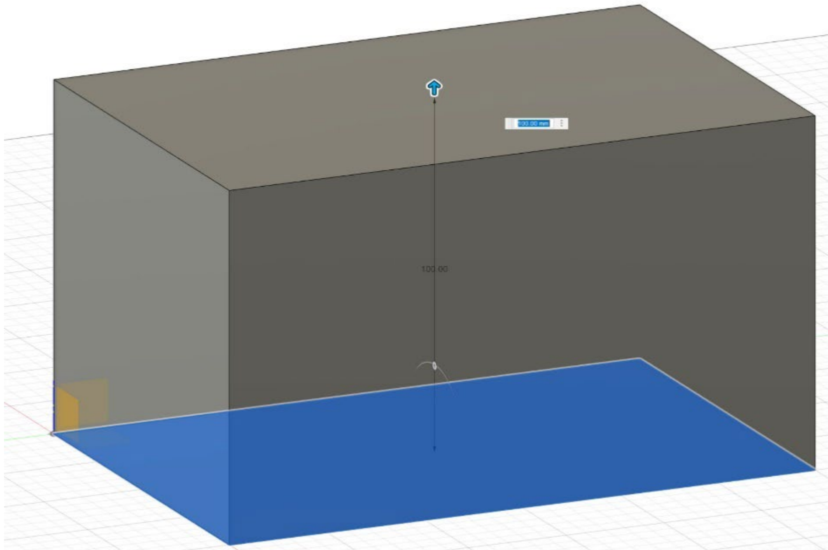
Now that you can see your sketch, press the "**Extrude**" icon**,** and click your sketch

This is what it should look like:





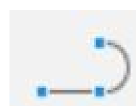
To now make the 2D shape into a 3D shape **bring the arrow up or type the size you want**. In this example, the rectangle is extruded to 100mm.

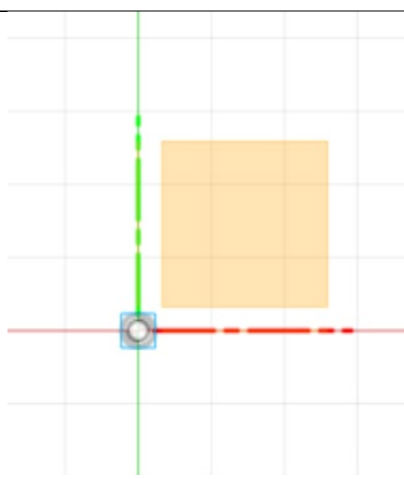


Congratulations, you have now extruded your shape!

EXERCISE 3: Making a triangle (any size)

For this example, click the plane you want to draw on and click the "**Create Sketch**" button. Make a [New Design]() and name it!

Start from the origin and click the "**Line**" button 

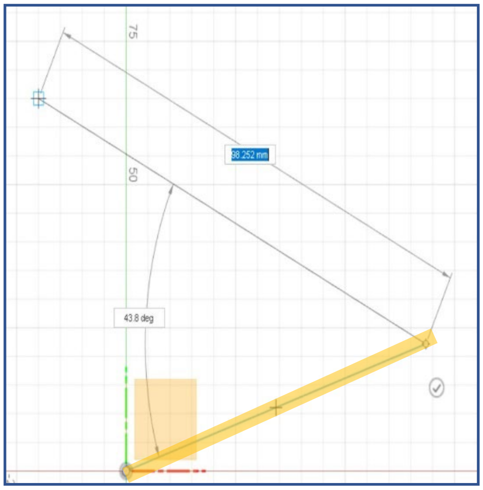
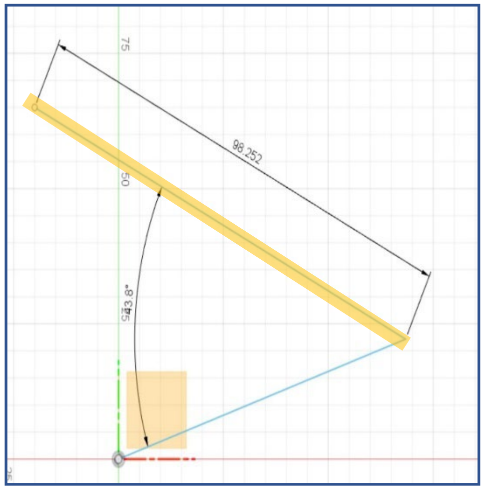
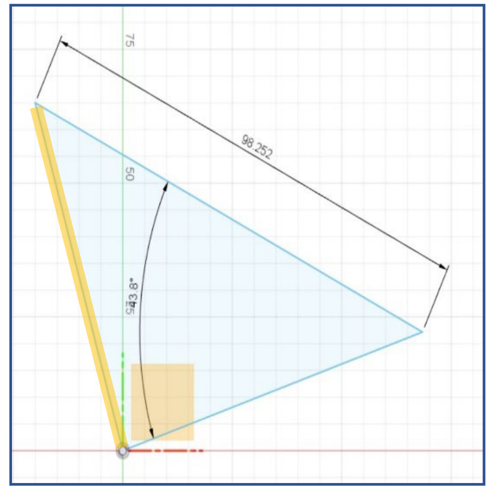
<table>
<tr>
<td>Origin: </td>
<td>
1. Start by clicking the origin point
2. Drag out your cursor to the desired length
3. Then drag out the next line to your size you want
4. Then connect the triangle with one last line

Be aware of the measurement size, Autodesk fusion shows the measurement and size in millimeters (mm)

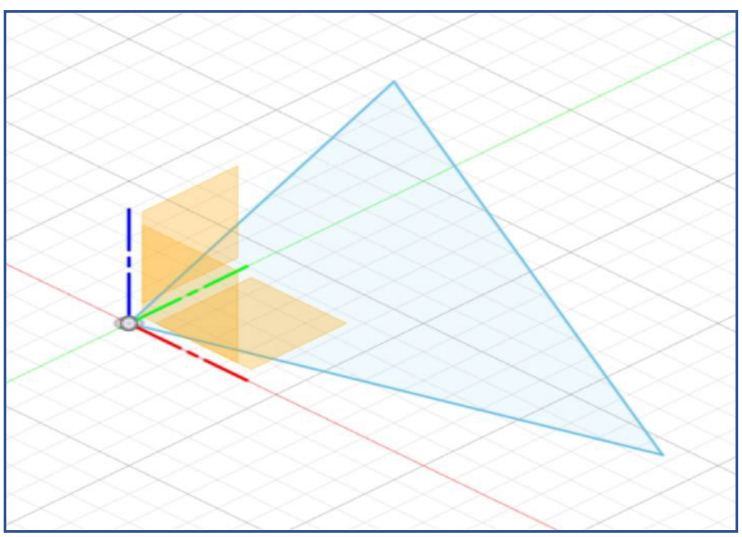
Hint: 1cm = 10mm
</td>
</tr>
</table>

Example of the steps:

1. Make the First line
2. Second Line
3. Third line

The Final product should look like any variation of a triangle that is fully connected:



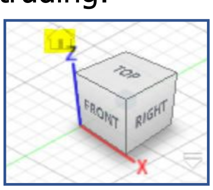
Once you have made this sketch, click Finish Sketch.

Now save your work by hitting the **Save Icon!**

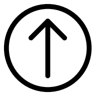
<u>Now we can extrude!</u>

EXTRUDING:

Before extruding:

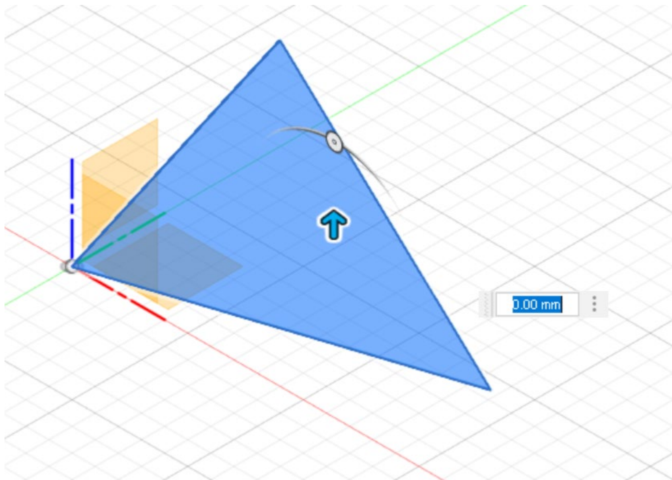


When out of sketch mode, **<u>always</u>** press the Home Button in the View Cube to properly view the 2D sketch. This is good practice.

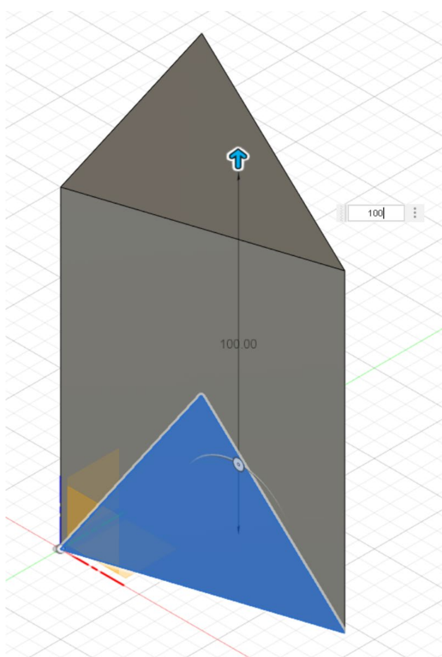


Now that you can see your sketch, press the **Extrude Icon,** and click your sketch

It should look like this:



To now make the 2D shape into a 3D shape **bring the arrow up** or **type the size you want**. In this example, the rectangle is extruded to 100mm



Congratulations, you have now extruded your shape!

HOMEWORK AND PRACTICE

After all you have learned, try to make these examples!

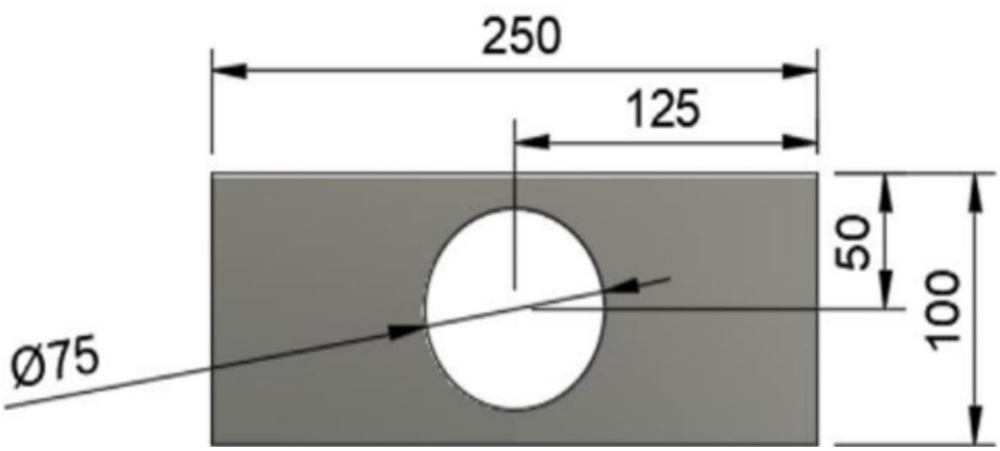
Checkmark them off when you have done them.

Refer to these exercises if you need help:

EXERCISES: CIRCLE RECTANGLE TRIANGLE

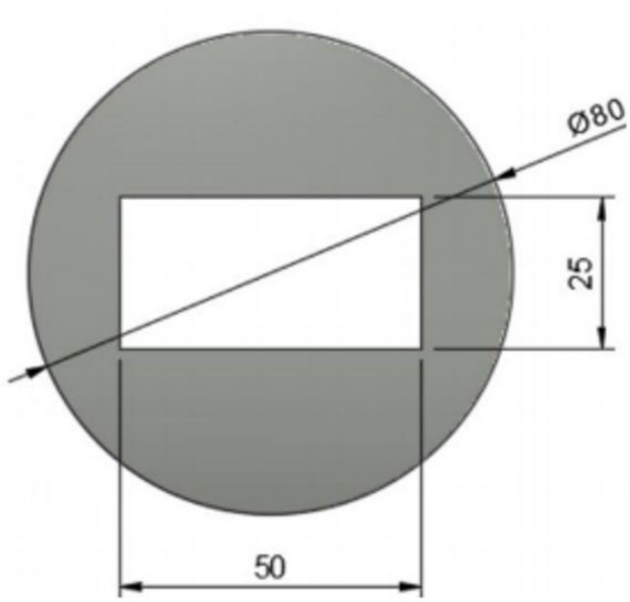
1. Create this object in Fusion 360 and extrude by 2 mm ☐

Exercise 1

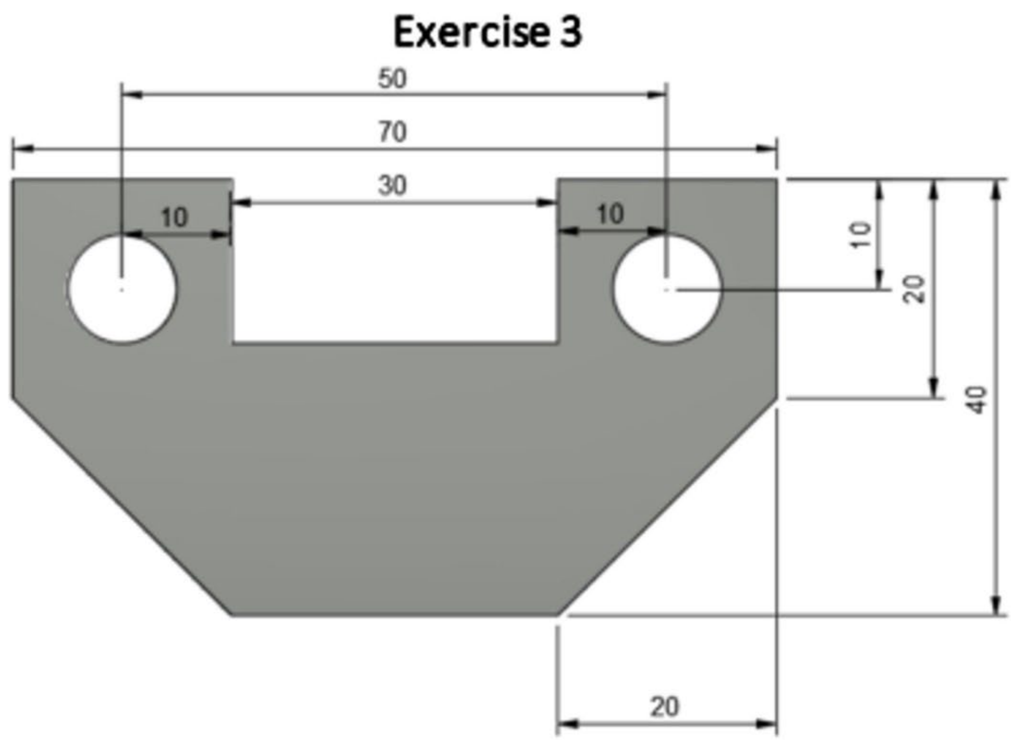


2. Create this object in Fusion 360 and extrude by 5 mm ☐

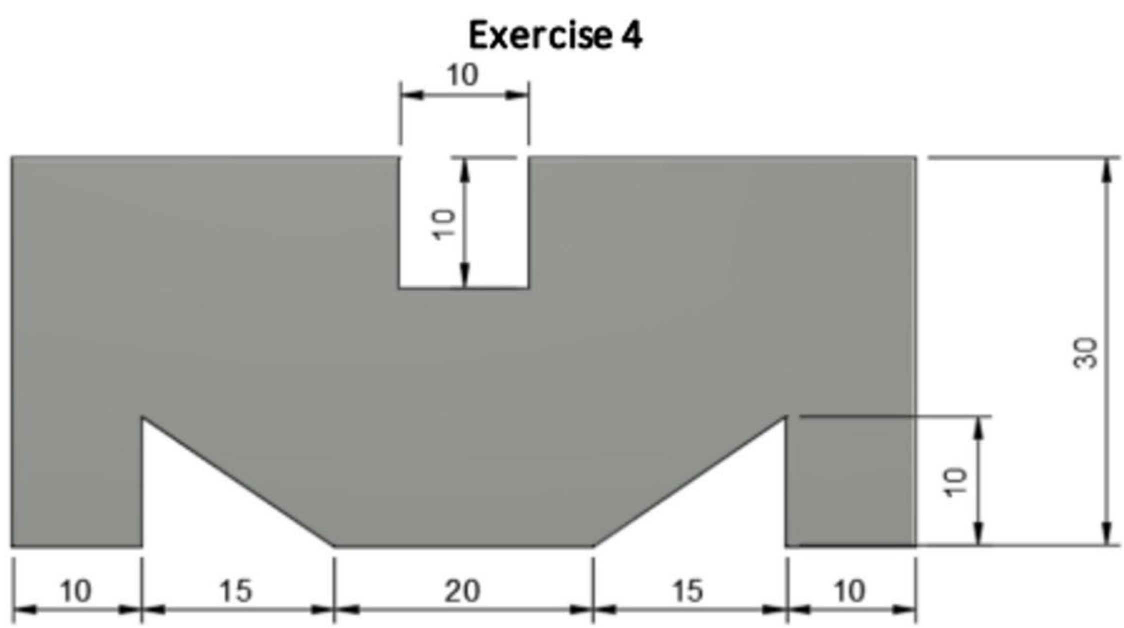
Exercise 2



3. Create this object in Fusion 360 and extrude by 7 mm ☐

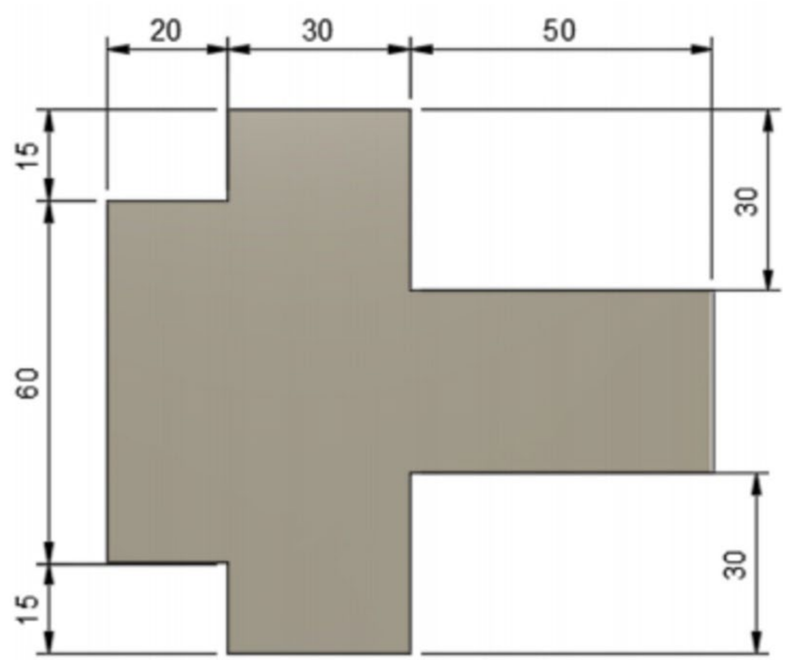


4. Create this object in Fusion 360 and extrude by 10 mm ☐



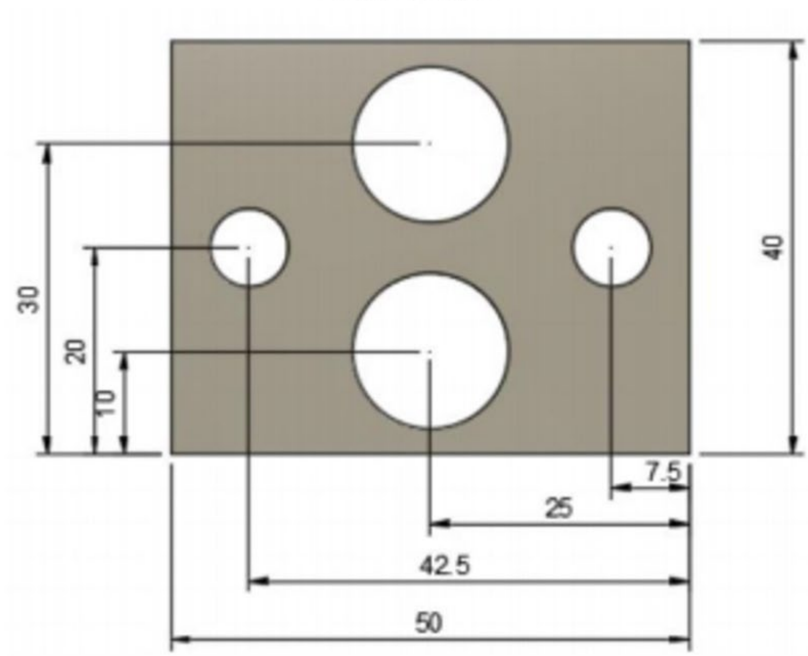
5. Create this object in Fusion 360 and extrude by <u>15 mm</u> ☐

Exercise 5



6. Create this object in Fusion 360 and extrude by <u>20 mm</u> ☐

Exercise 6

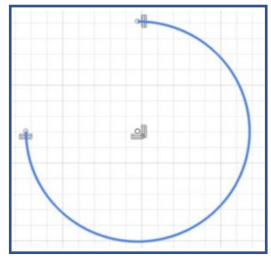


Good job if you have completed all these exercises!

2: ADVANCED SKETCHING

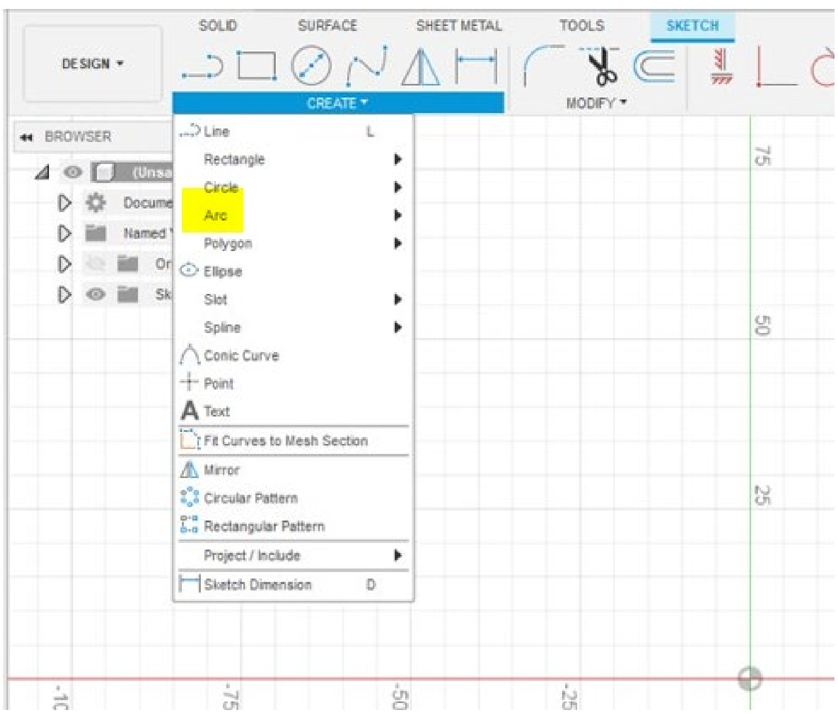
INTRODUCTION TO ARCS

Arcs are used to draw segments of a circle. Their properties are when sketching is a radius/ diameter, an arc center and a height/width.

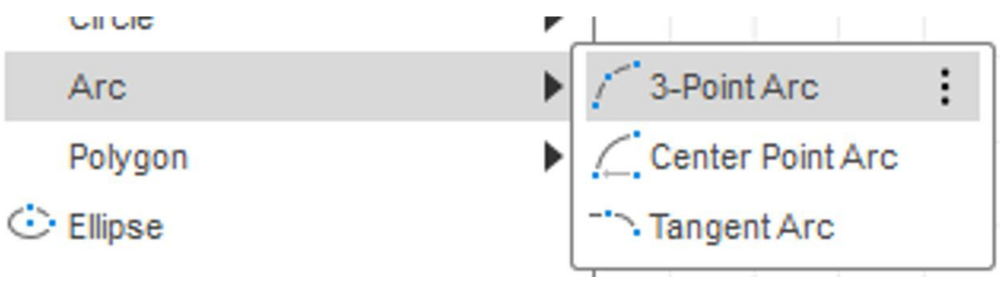


How to create an Arc:

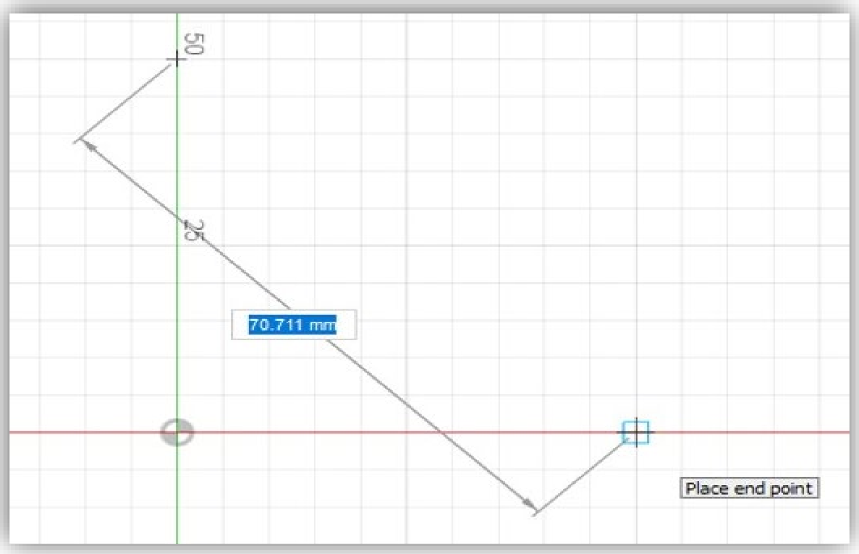
1. Under "Create" select "**Arc**"



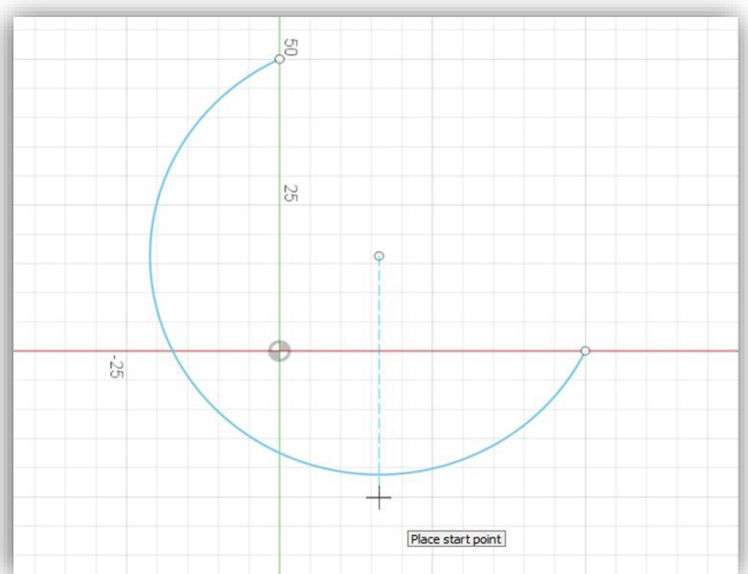
2. Select what Arc you want, in this example we want the "**3-Point Arc**"



3. Select two points and insert your desired measurements.



4. Adjust your Arc accordingly.



5. Complete the sketch by pressing finish sketch on the top right.

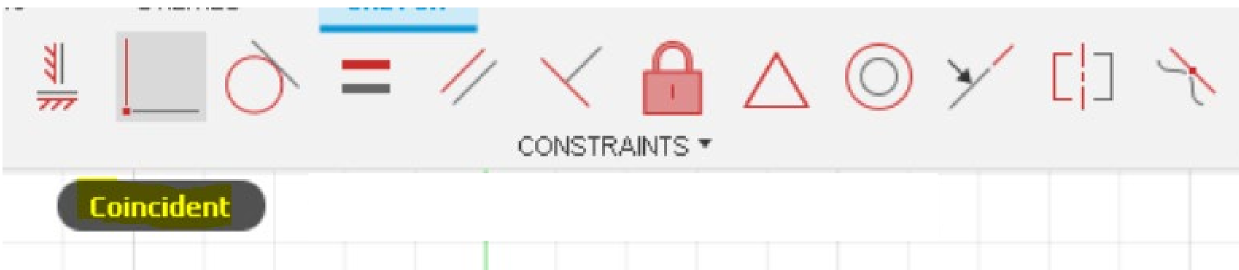
INTRODUCTION TO CONSTRAINTS

Constraints are used to **join two points** (not lines) together. There are different types of constraints. Such as Coincident, Equal, Horizontal/vertical and Concentric.

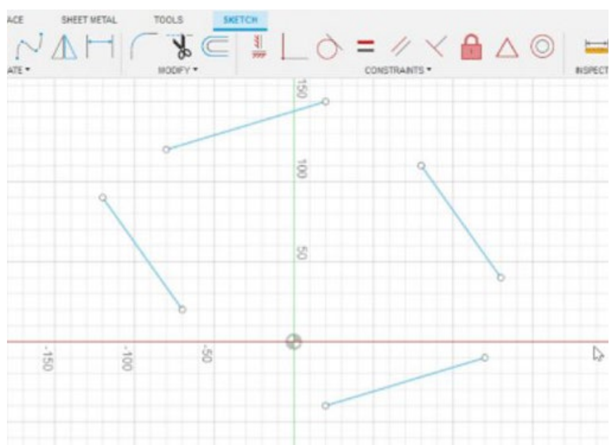
How to use each type of constraint:

COINCIDENT

1. Click on "**Coincident**" in the constraints section of sketch tools.

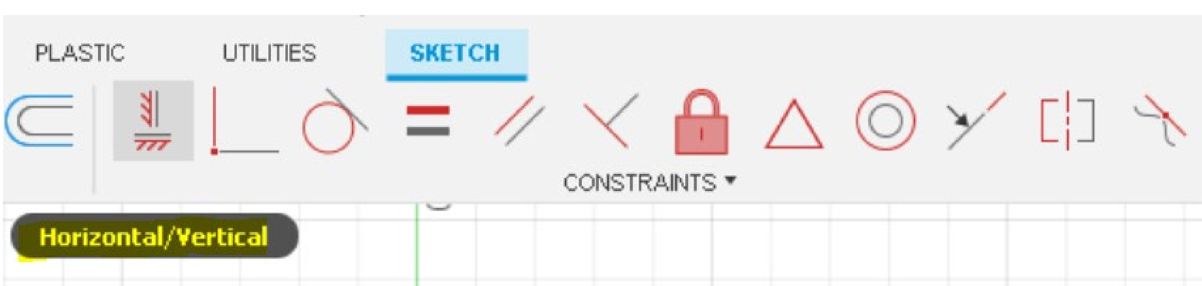


2. Then select the two points you want to join together

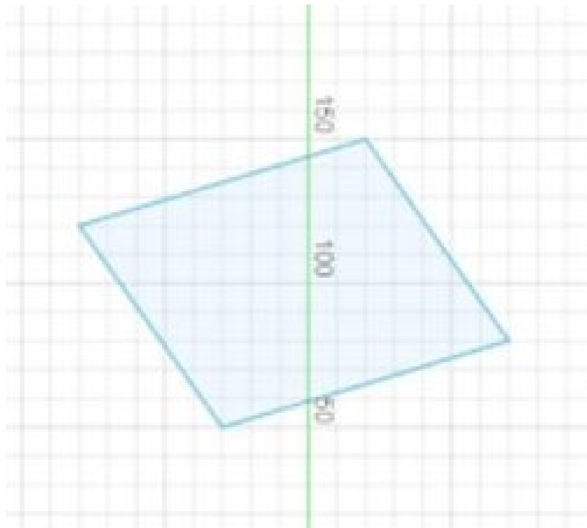


HORIZONTAL AND VERTICAL (on edges)

1. Click on "**Horizonal/Vertical**" in the constraints section of sketch tools.

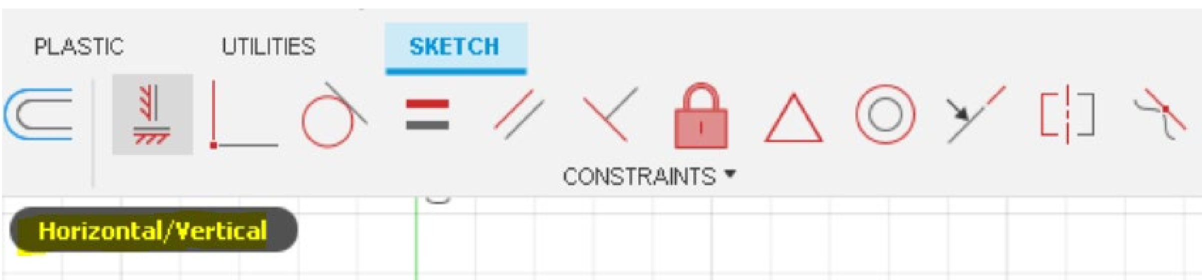


2. Then select the <u>edges</u> you want to join together.

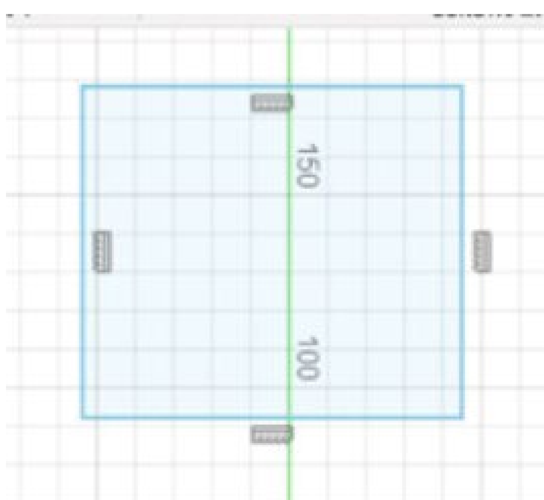


HORIZONTAL AND VERTICAL (on points)

1. Click on "**Horizonal/Vertical**" in the constraints section of sketch tools.

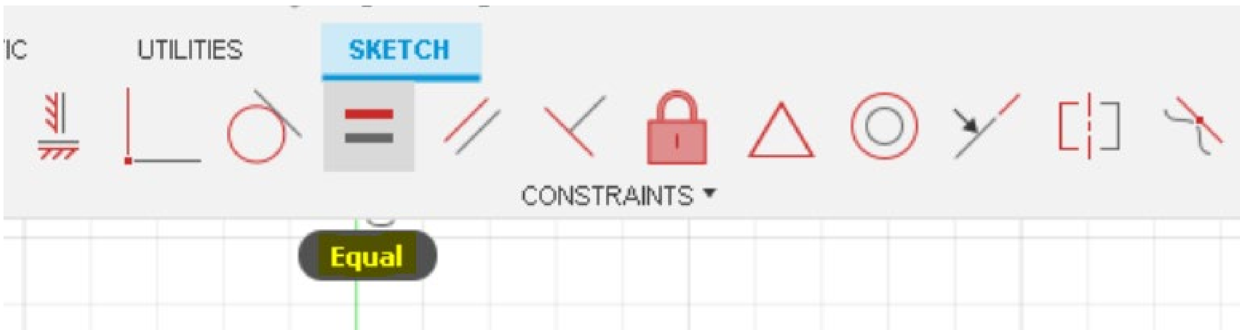


2. Then select the **points** you want to join together.

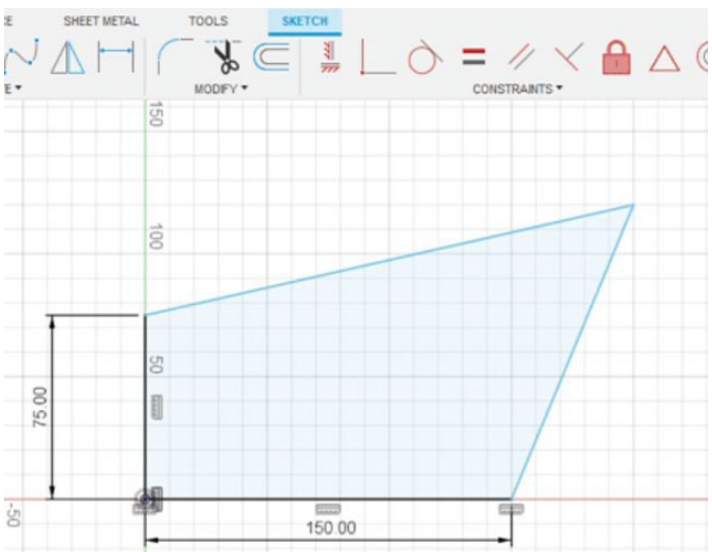


EQUAL

1. Click on "**Equal**" in the constraints section of sketch tools.



2. Then select the **lines** you want to join together.

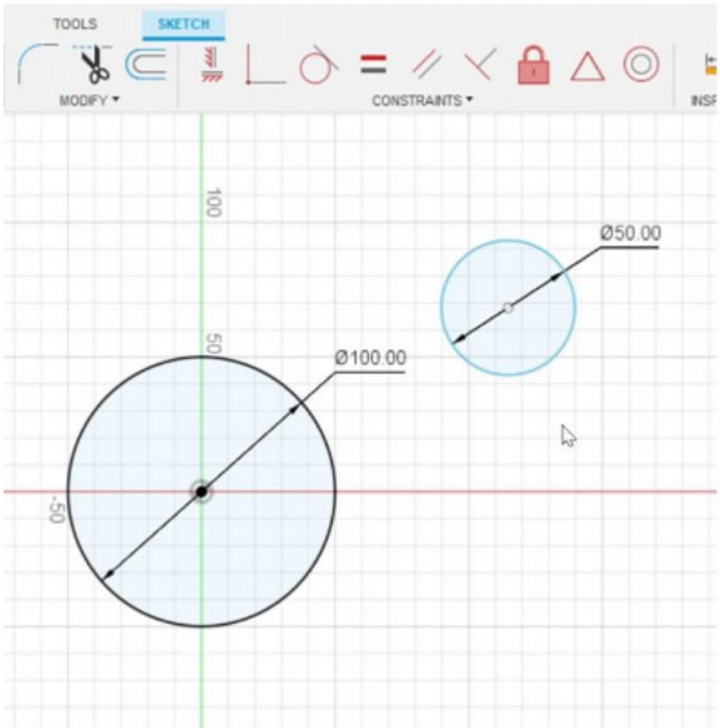


CONCENTRIC

This constraint is usually **used only on arc or circles**. It makes the center of the two entities selected to be the same.

1. Click on "**Concentric**" in the constraints section of sketch tools.

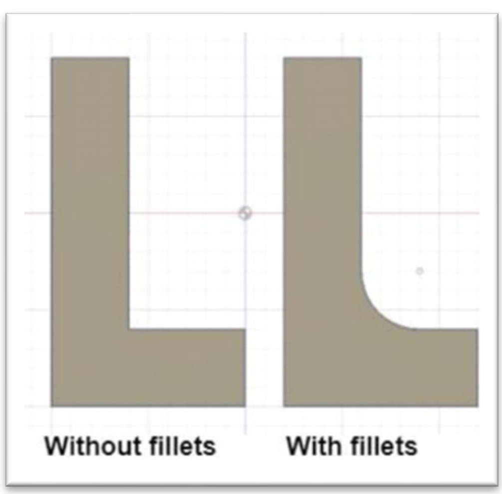
2. Then select the first entity and then the second entity.



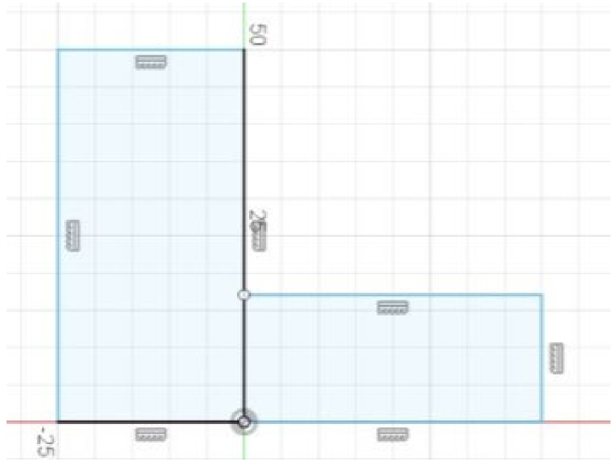
INTRODUCTION TO FILLETS

Fillet round corners to form a radius, Fillets are rounded to round sharp corners, **Sharp corners are considered weak points** (i.e., first points to break) in a design.

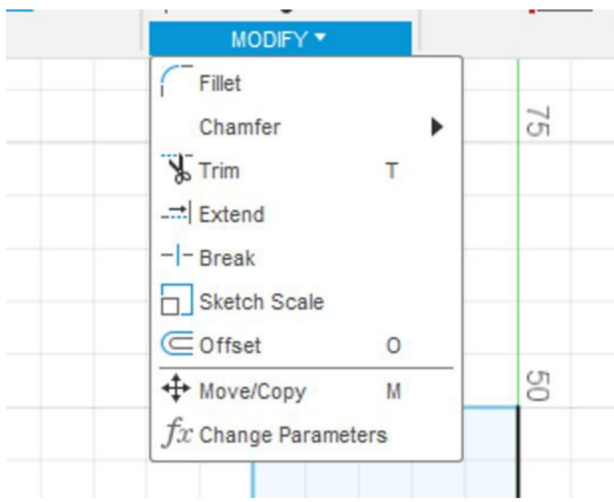
Therefore, using fillets in your design will help avoid sharp corners in your design which you should always do.



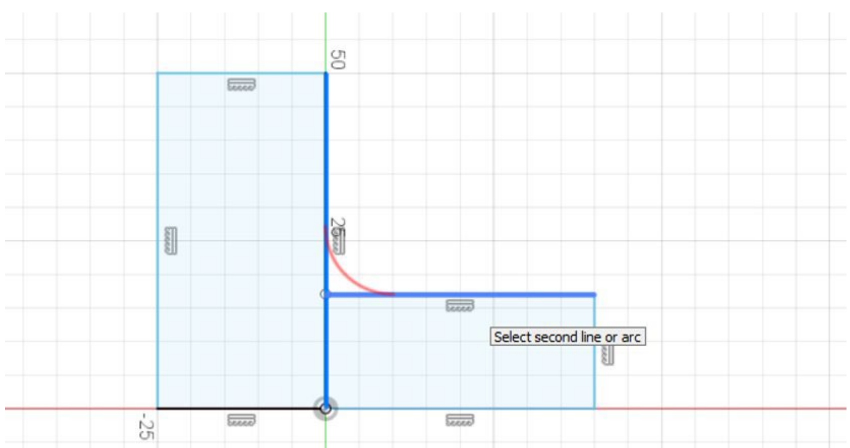
1. Select your shape/object you want filleted.



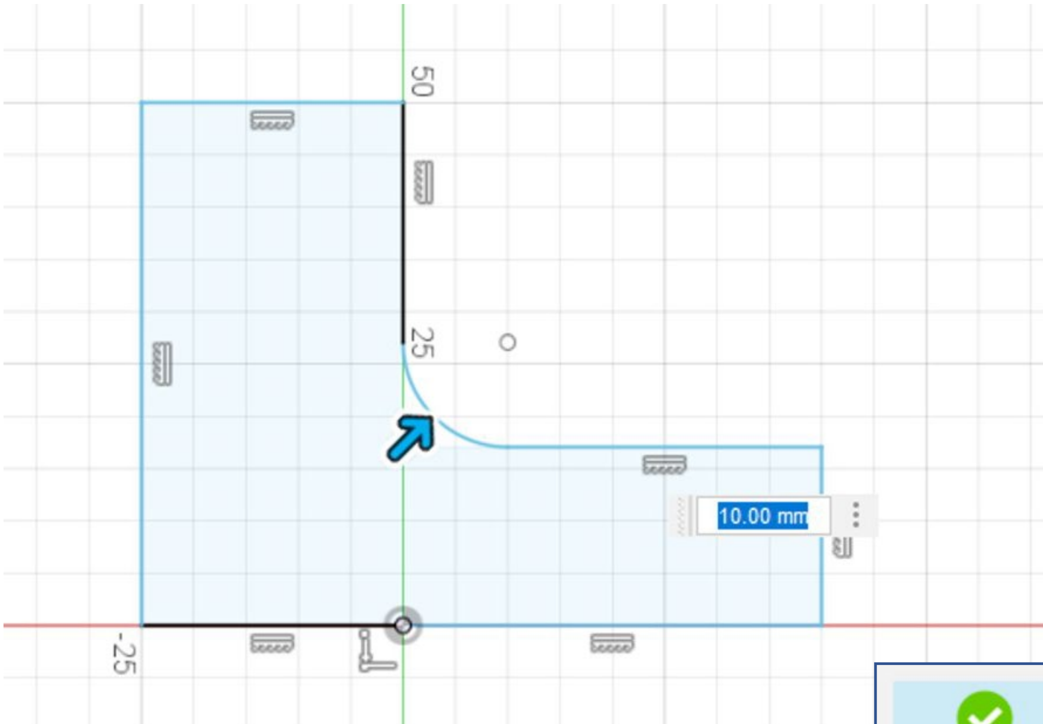
2. Under modify select “**Fillet**”



3. Select the **two lines** you want to fillet

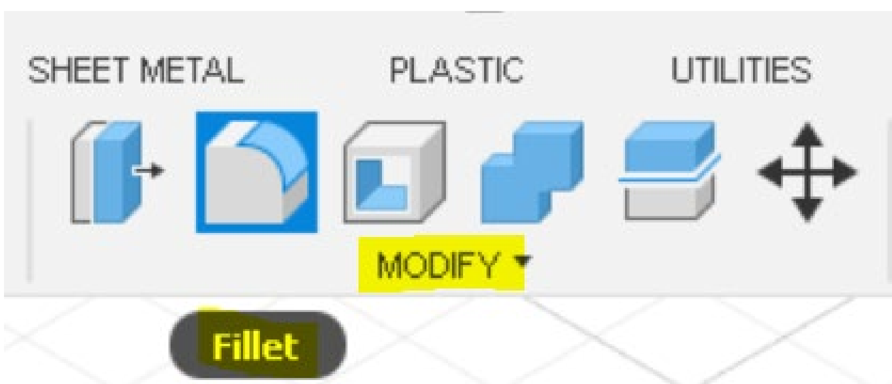


4. Input your desired measurements



5. Complete the sketch by pressing “**Finish Sketch**” on the top ri

USING FILLETS AFTER A SKETCH IS EXTRUDED



You can extrude fillets on an extruded part, by **pressing the Fillet button** above Modify and **click the edge you want to smooth**.

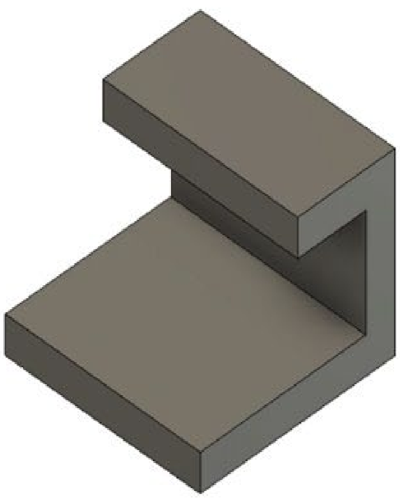
<u>Use the arrow to adjust how far you want to file your shape</u>. Once you file it to your desired sized, you can press enter and you will have a smooth part!

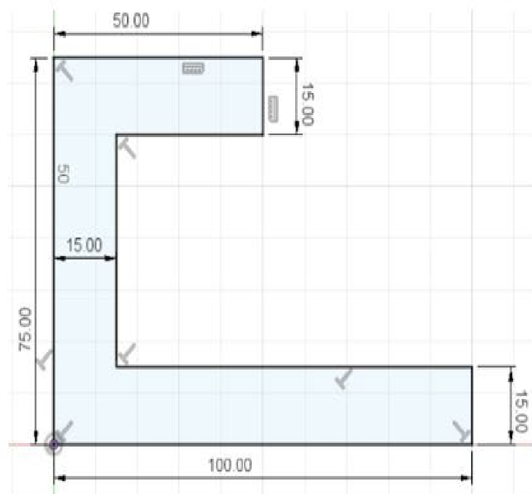
HOMEWORK AND PRACTICE

After all you have learned, try to make these examples!

Checkmark them off when you have done them.

Exercise 1: Constraints ☐

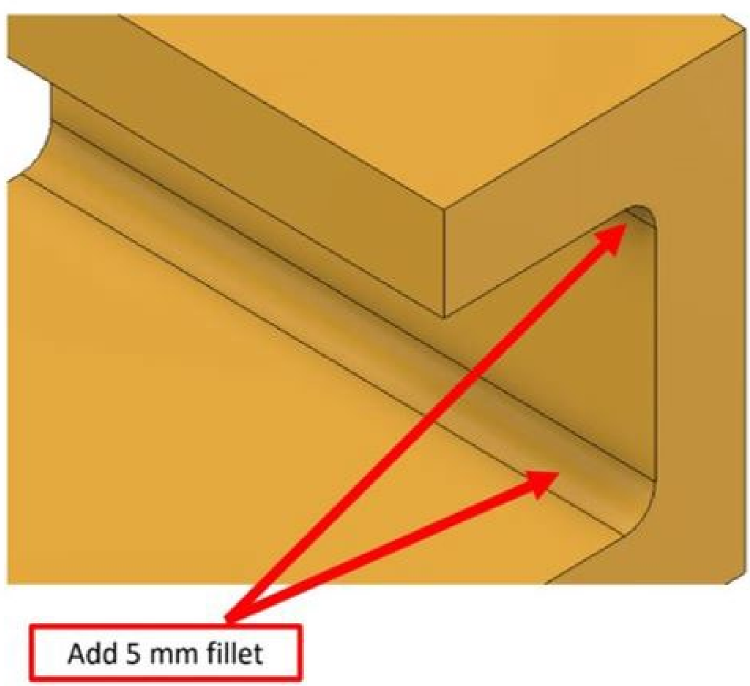




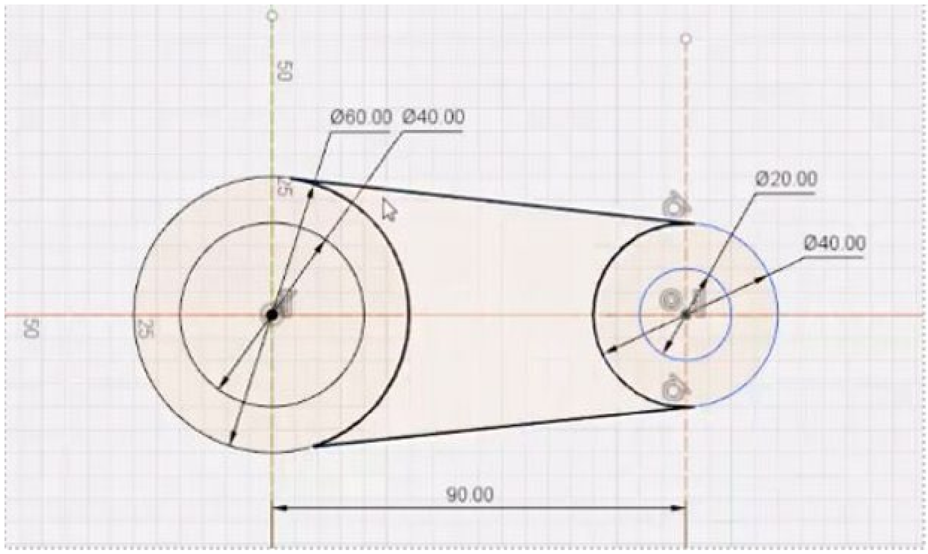
Attempt this shape using constraints, Extrude the component by 100mm

Extension: Internal fillets

- Create two internal 5mm fillets



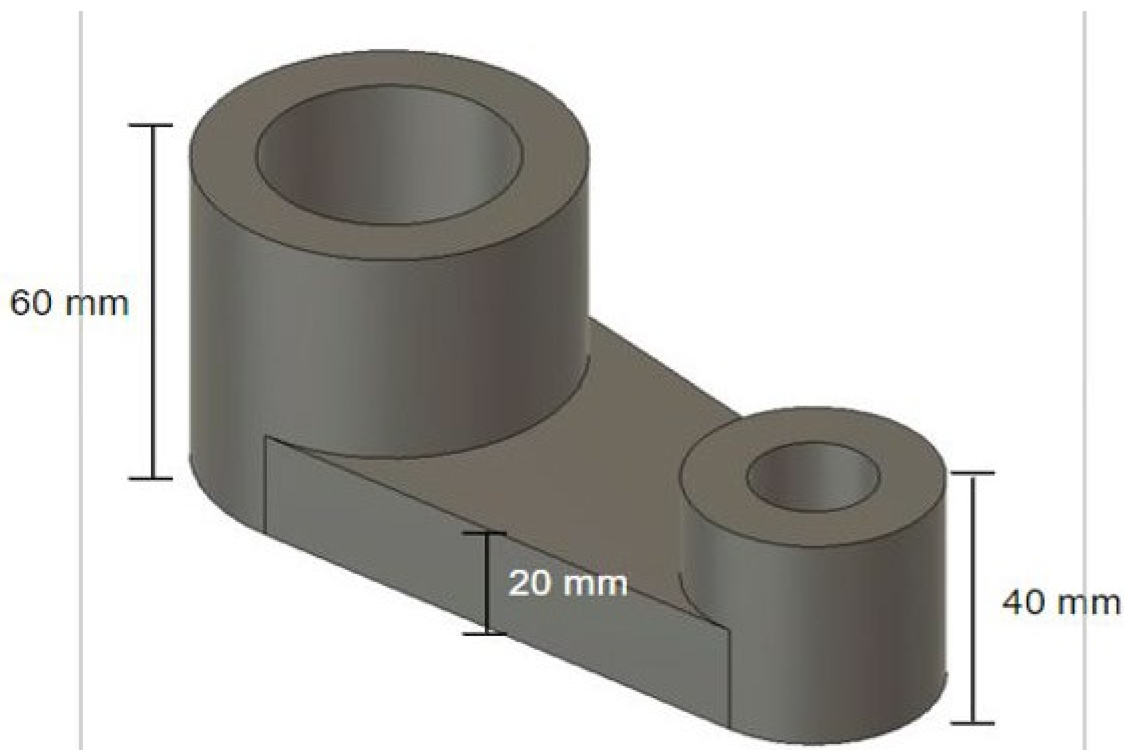
Exercise 2: ☐



Hint: Use Concentric Constraints!

Extension:

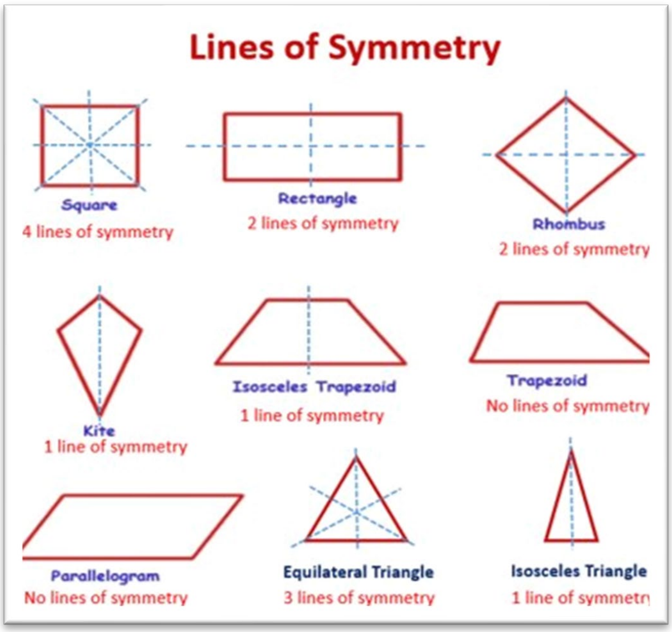
- Extrude to the following dimensions:



3: MIRRORS AND PATTERNS

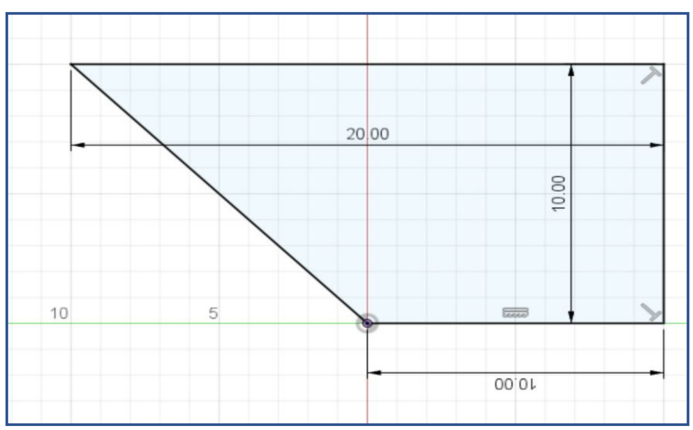
MIRRORING TOOL

The mirroring tool can be used to **reflect sketched entities**. When using the mirror tool, you need to identify and create a mirror line. The mirror line is a line that cuts your sketch into two equals but reflected parts. You can imagine that the mirror line is the line of symmetry.



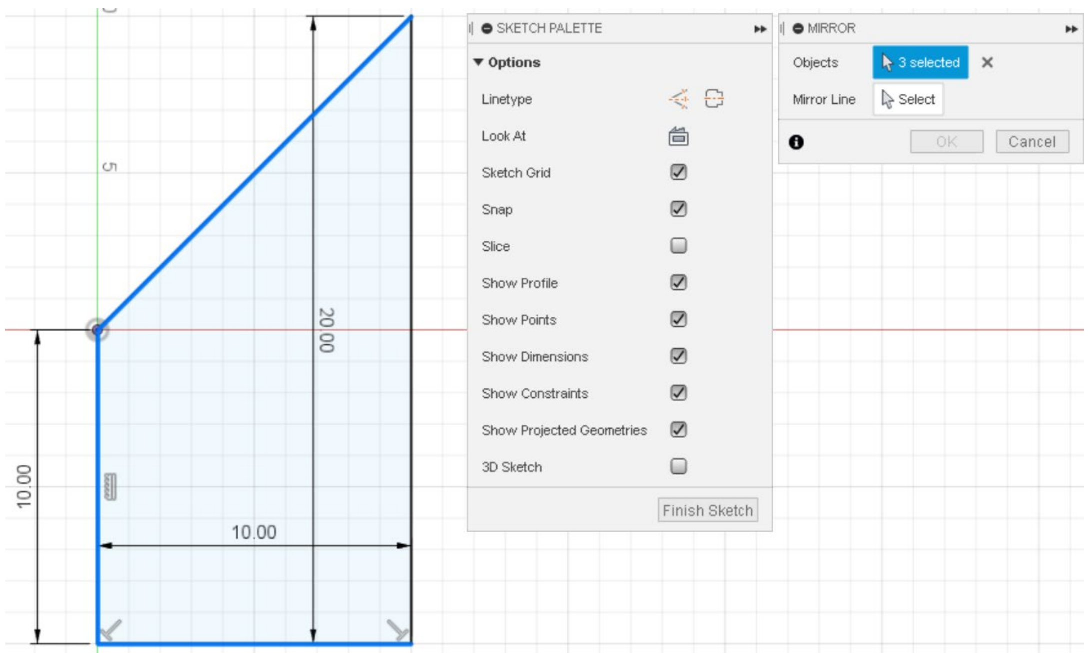
HOW TO USE THE MIRROR TOOL

For this example, we will use this drawing. Create this sketch by using the line tool. Create the 4 lines as shown.

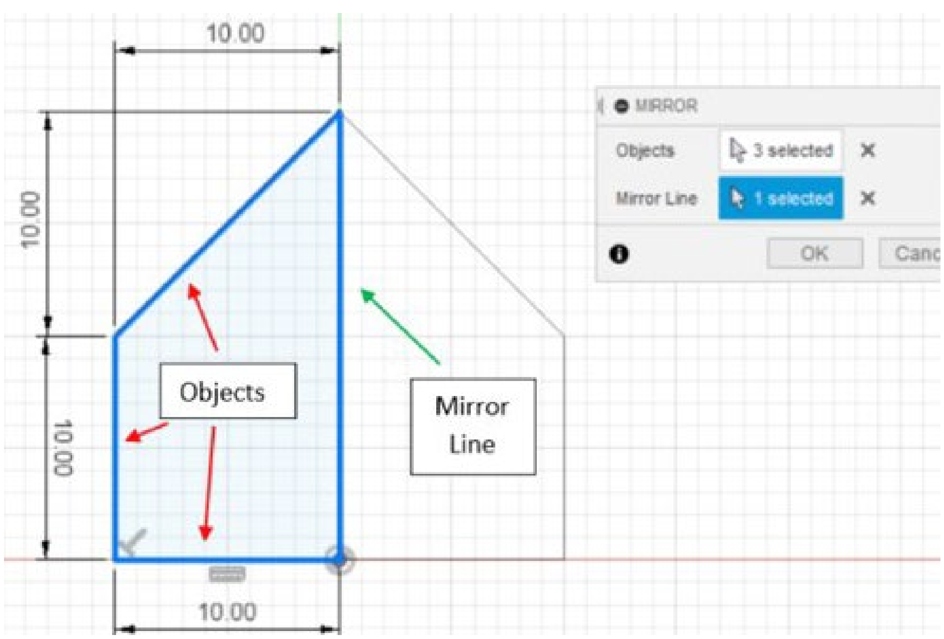


1. After you have created your sketch, select the "**Mirror**" tool.

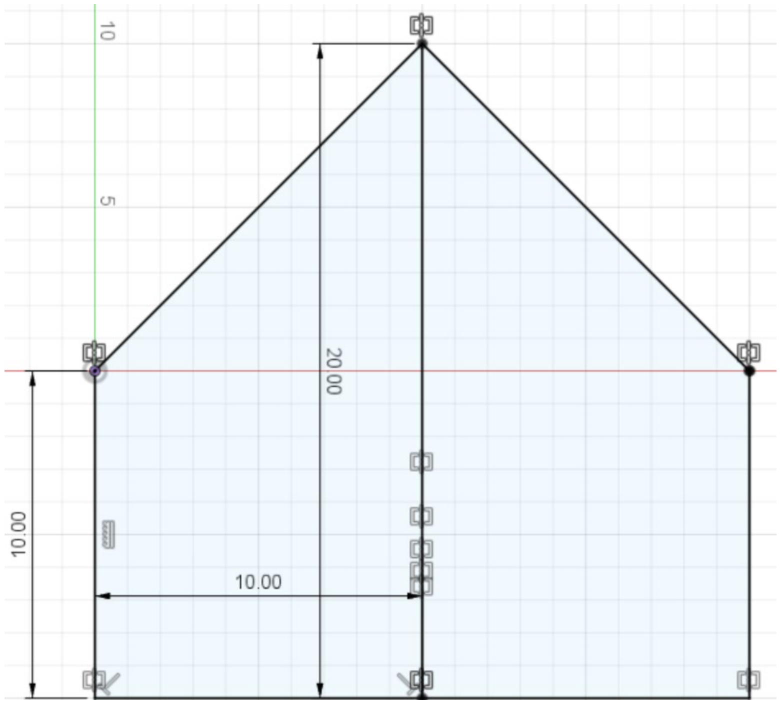
2. First select your objects by making sure you are on the "**Objects**" tab. When you want to select an object, just click the line and it will highlight them blue.



3. Next, select your mirror line by switching to the "**Mirror line**" tab and selecting the line you want to be your line of symmetry.



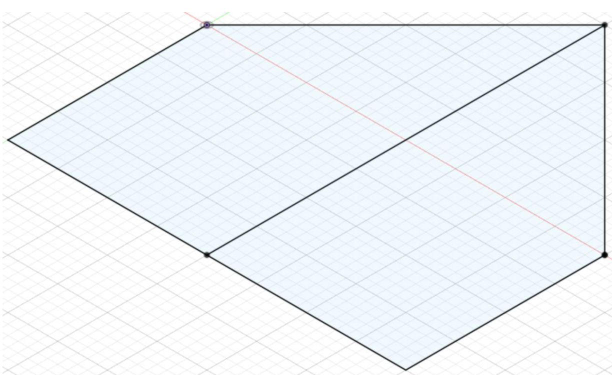
4. Once you have selected everything, click "**OK**" and watch your sketch mirror!

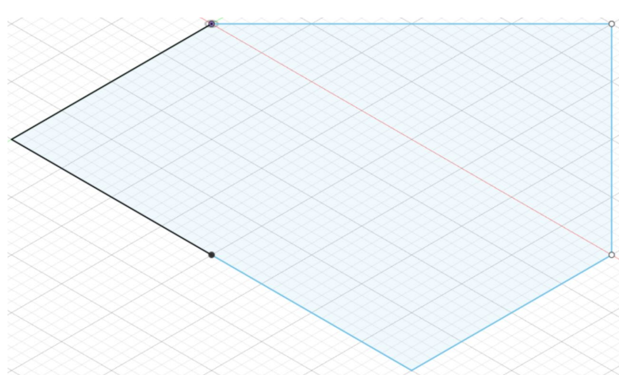


EXTENSION:

When extruding you sketch, you don't want a line separating your face. **You can delete the mirror line to make one face!**

Mirror line VS No Mirror Line





You can see the difference between how many faces you can extrude all together. With the mirror line, you can extrude 2 different parts. Without the mirror line, you can extrude 1 part. It is important to remember that the **mirror line can separate your sketch into different parts when you might only want 1 part**.

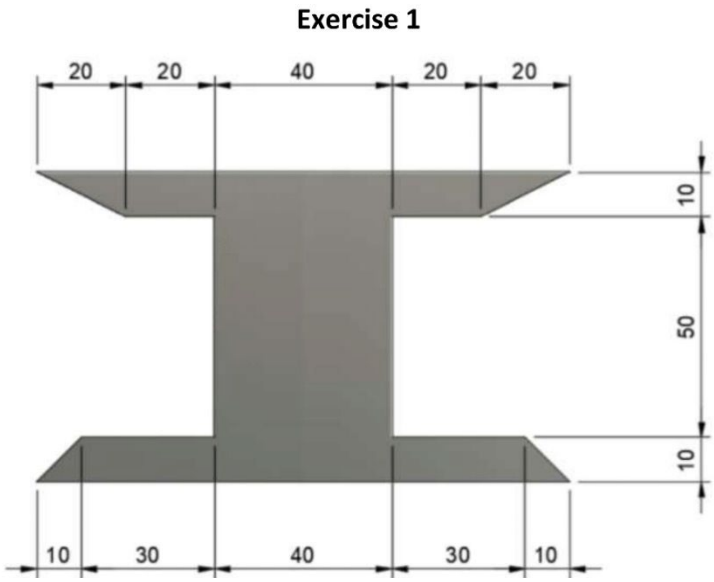
MIRRORING PRACTICE

After all you have learned, try to make these examples!

<u>Checkmark them off when you have done them.</u>

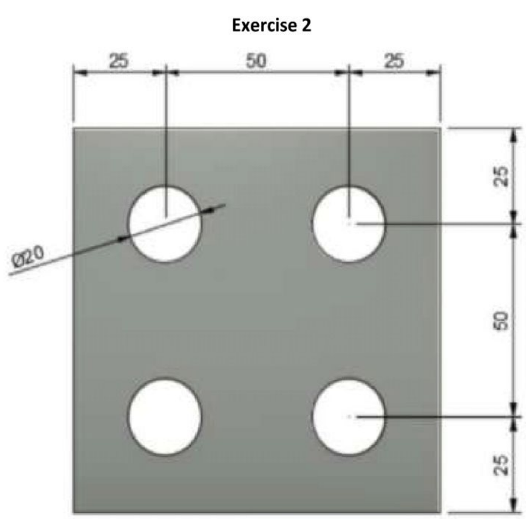
Exercise 1: ☐

Attempt this sketch by applying the mirror tool, Extrude the sketch by <u>10mm</u>



Exercise 2: ☐

Attempt this sketch by applying the mirror tool, Extrude the sketch by <u>10mm</u>



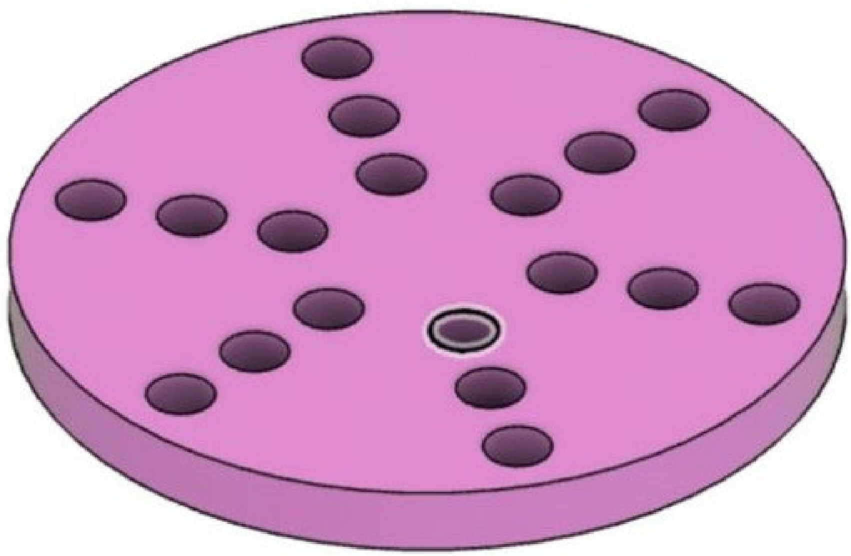
Hint: Find the lines of symmetry to sketch these exercises faster!

PATTERN TOOL

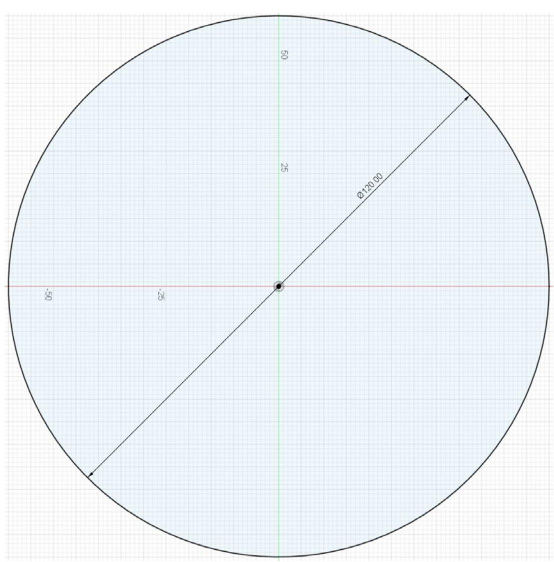
The pattern tool **allows the same shape to be repeated** because sketching each shape individually would take a lot of time. Mirrors and the usage of constraints may also work but patterns are far more efficient time wise. In Fusion360, we can use the pattern tool to sketch entities either in a circle or a rectangle.

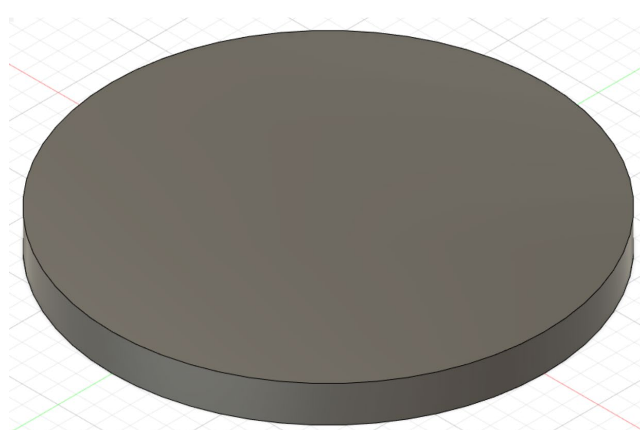
PATTERN TOOL (CIRCLE)

In this example we are going to create this part:

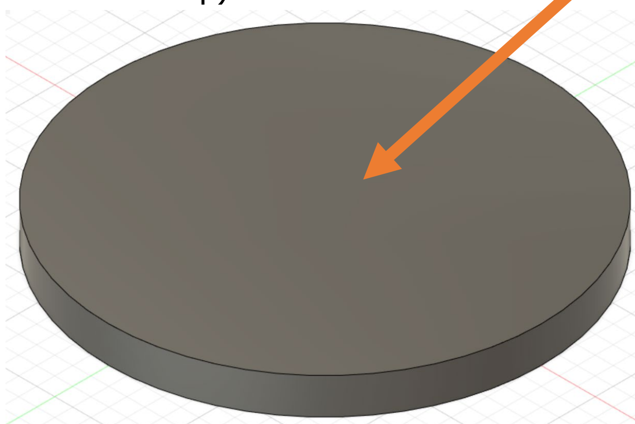


1. Sketch a circle on the bottom plane that is **120mm/12cm** and extrude by **10mm**

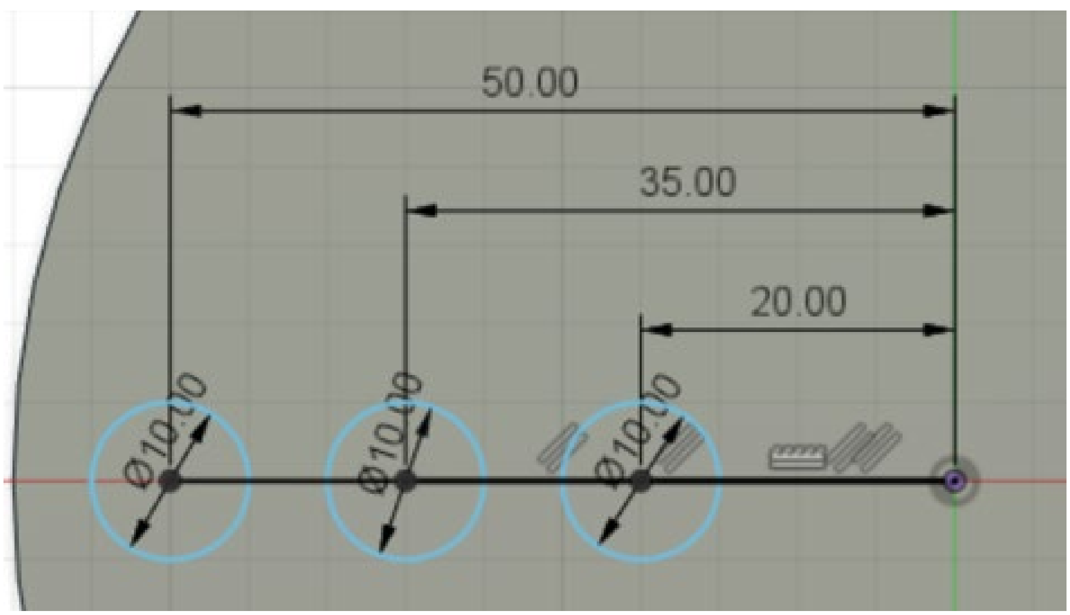




2. Create a sketch on the top surface of the circle. (**Select the top face** and the click "**Create Sketch**" at the top)

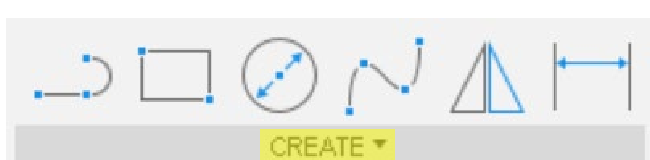


3. Create 3 new circles, all with a **diameter of 10mm**. Make sure the circles are <u>horizontal to the origin</u> and separated like this:

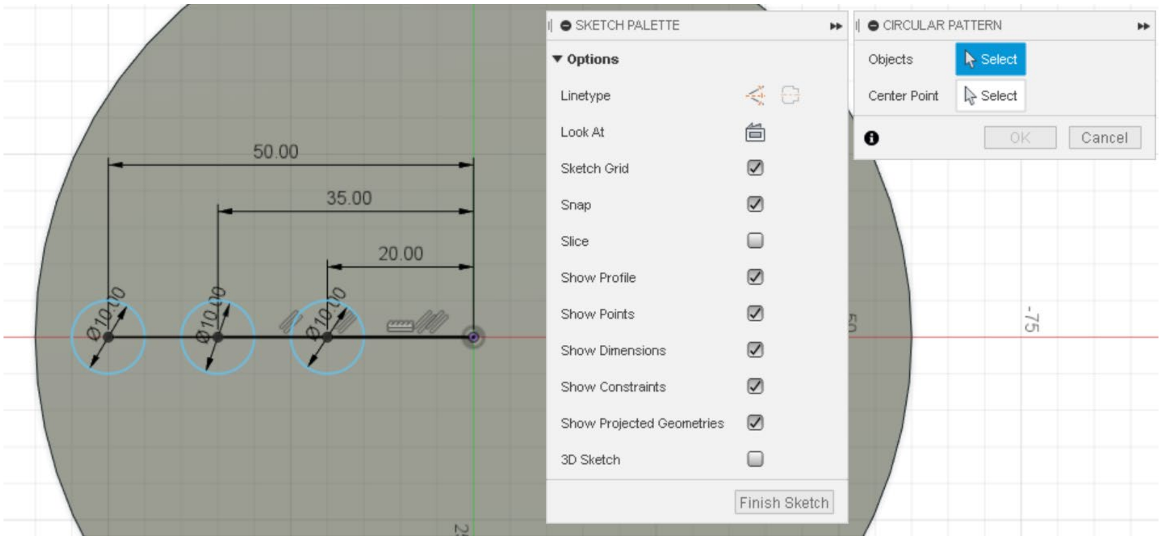


HINT: Add a Horizontal/Vertical constraints between the origin and center of the circles

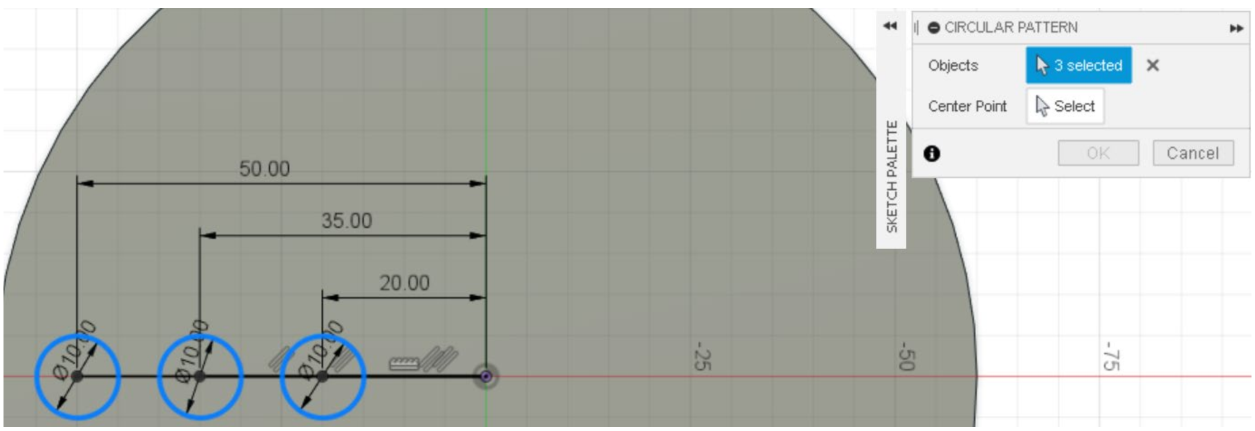
4. Now click Create and scroll down to "**Circular Pattern**" and click it.



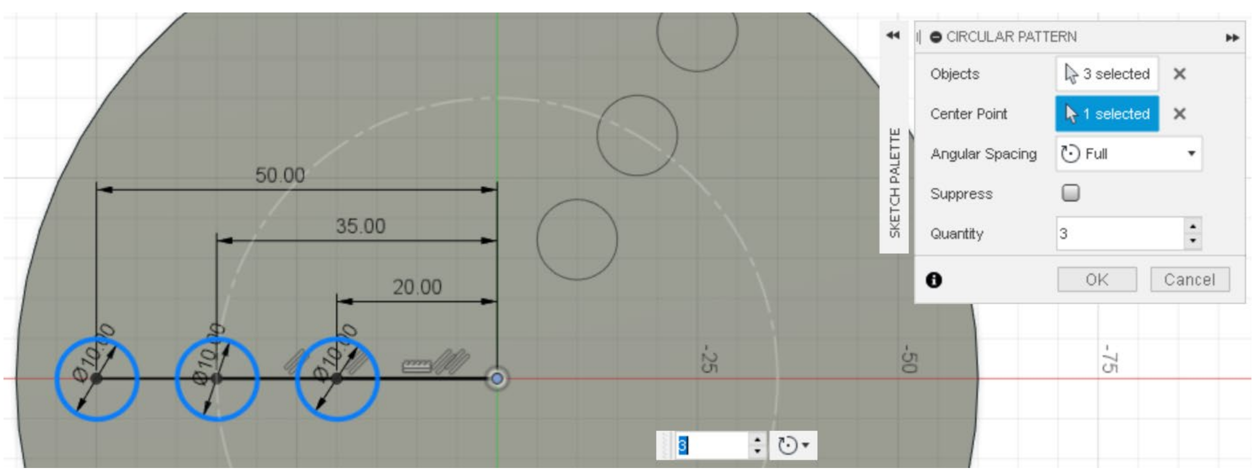
It should look like this:



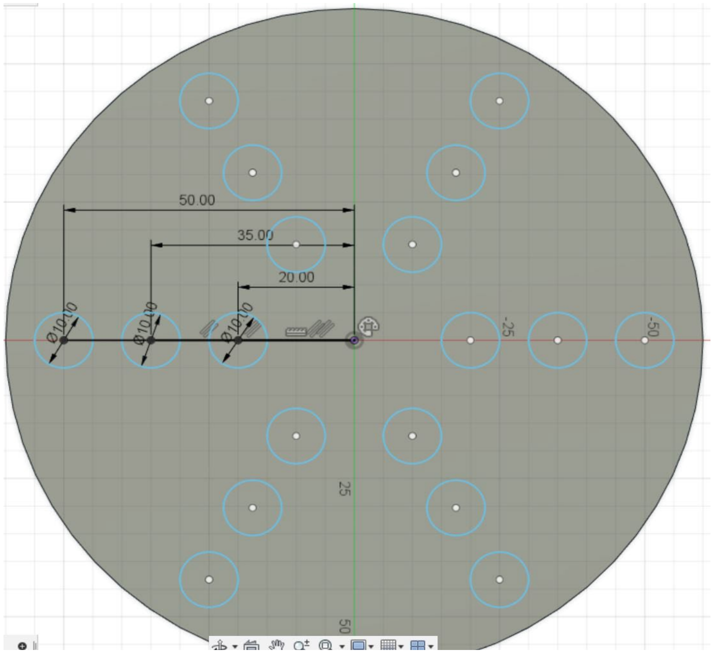
5. Make sure you are under the "**Objects**" tab and select the 3 circles you sketched.



6. Now switch to the "**Center Point**" and select the origin



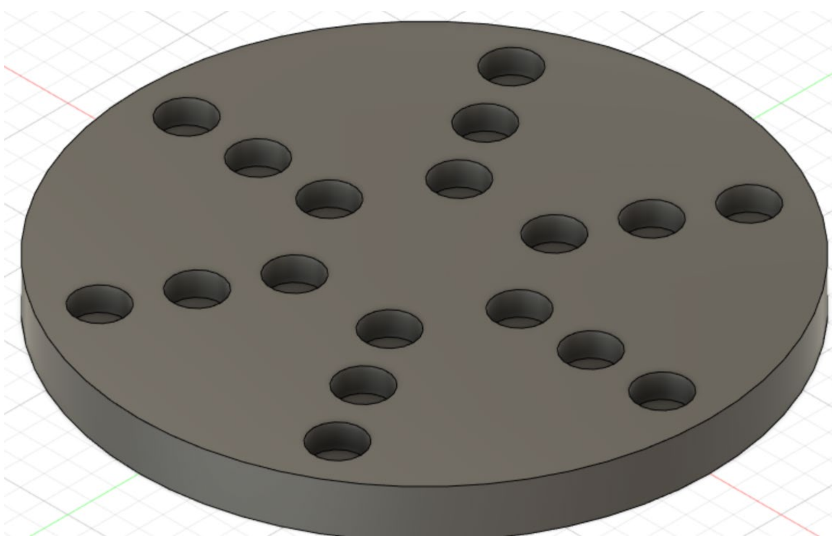
7. Decide how many numbers of copies we need, for this example we need 6. Change the "**Quantity**" to **6**, click OK.



It should now look like this!

8. Click "**Finish Sketch**"

9. Select all the circles and extrude by <u>-5mm</u>. Final product should look like this:



You can now see that you have created a pattern on this shape!

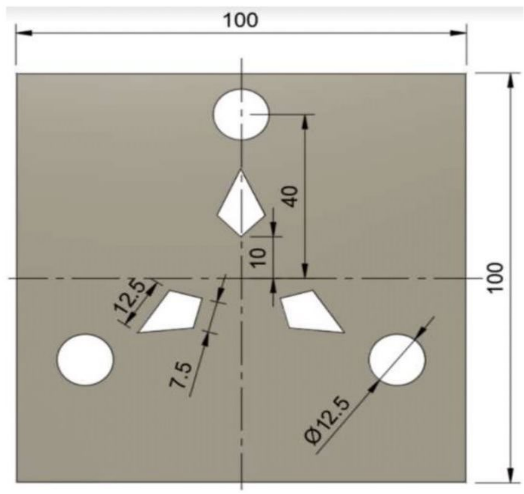
PATTERN TOOL PRACTICE

After all you have learned, try to make these examples!

<u>Checkmark them off when you have done them.</u>

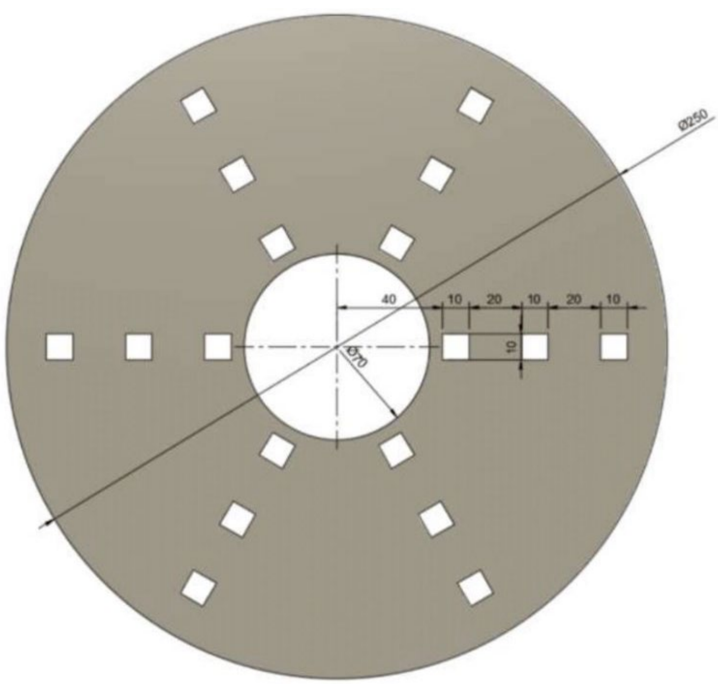
Exercise 1 ☐

Attempt this sketch by applying the pattern tool, Extrude the sketch by <u>10mm</u>



Exercise 2 ☐

Attempt this sketch by applying the pattern tool, Extrude the sketch by <u>10mm</u>



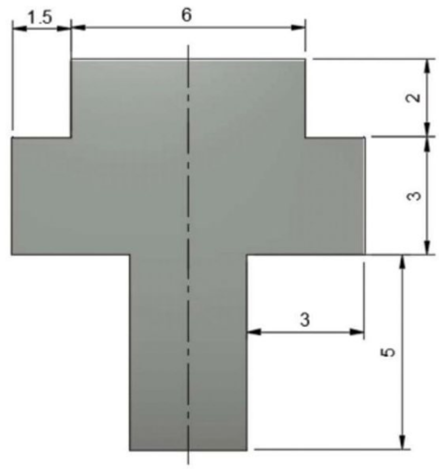
HOMEWORK AND PRACTICE

After all you have learned, try to make these examples!

Checkmark them off when you have done them.

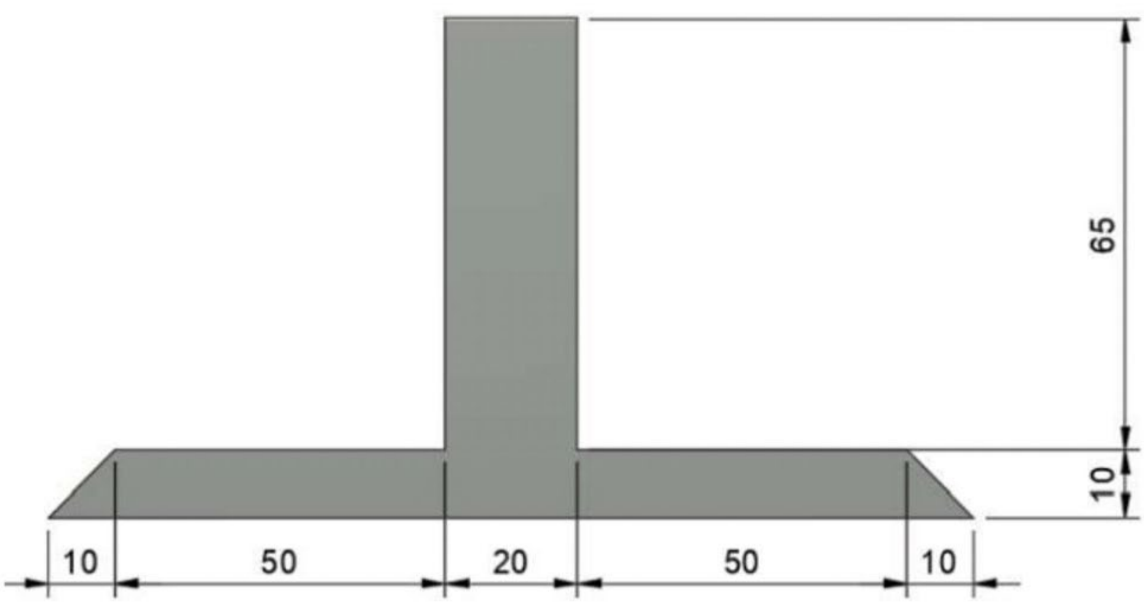
Exercise 1 ☐

Attempt this sketch by applying the either the mirror or pattern tool, Extrude the sketch by 12mm



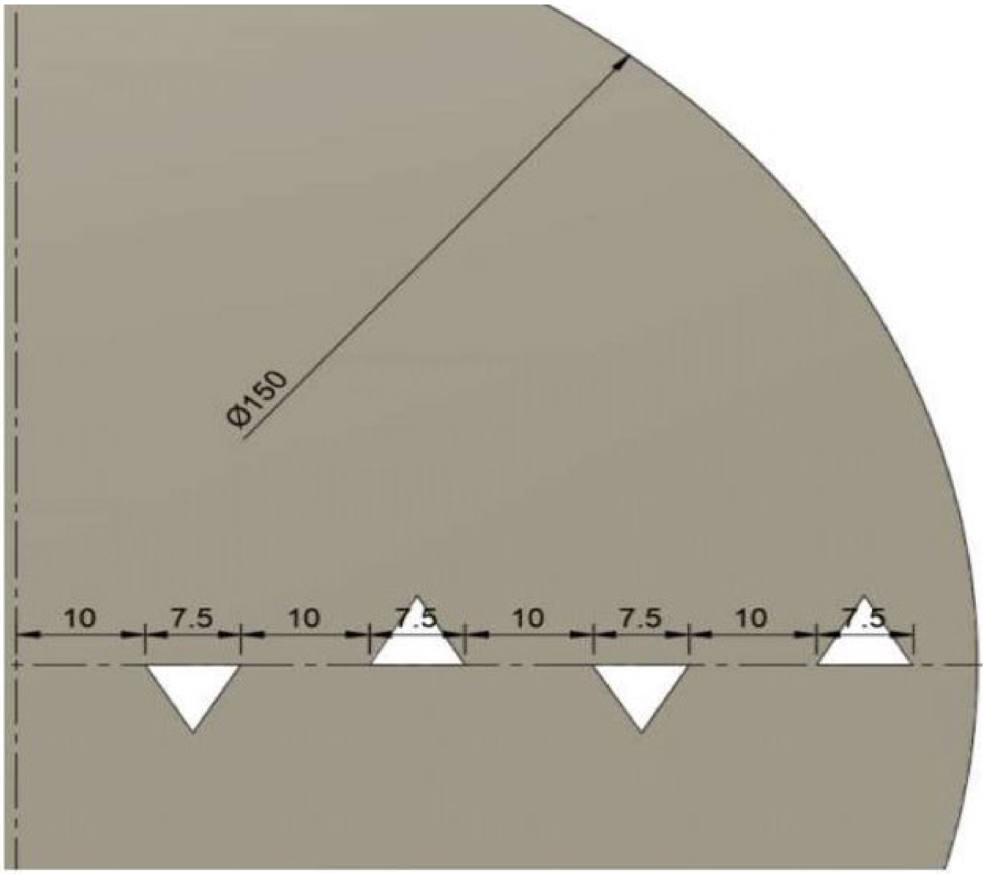
Exercise 2 ☐

Attempt this sketch by applying the either the mirror or pattern tool, Extrude the sketch by 12mm

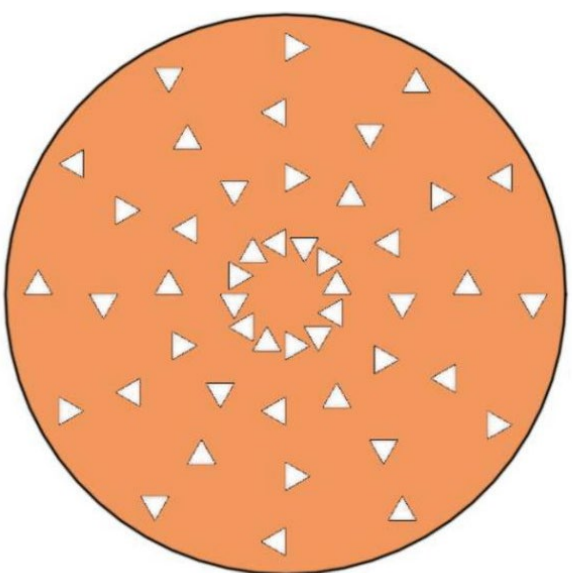


Exercise 3 ☐

Attempt this sketch by applying the either the mirror or pattern tool, Extrude the sketch by <u>12mm</u>



Final Product:



4: REVOLVE TOOL

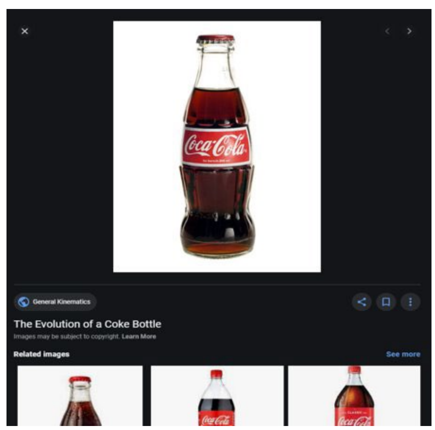
Not all 3D shapes can be created by using the extrude tool. Think of a soccer ball or vase, can you design it using extrude? Here is where we **use the revolve tool to extrude circular and round shapes**. The revolve tool allows us to take a sketch and revolve it around an axis of revolution to create 3D object.

For this example, we will try to create this Coca Cola bottle on Fusion360

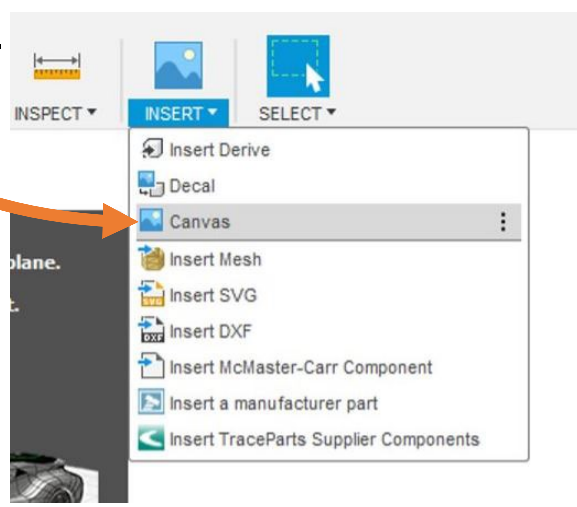
- For this we are going trace the profile of the bottle and then revolve it.
- In order to trace the profile, we need to import an image of a bottle.

IMPORTING IMAGES

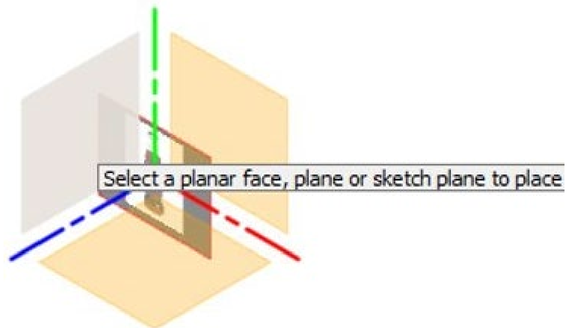
1. Find an image of a coke bottle and procced to download it as an image



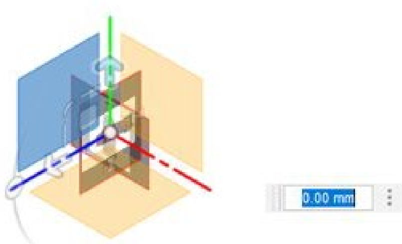
2. Go to insert. Then "**Canvas**".



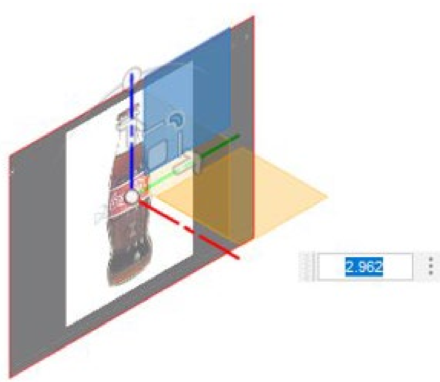
3. For the Face select the **left plane**



4. For "**Image**" select your downloaded image of the coke bottle



5. Scale your image to your desired size



REVOLVING WITH SKETCHES

CLASS VIDEO:

Watch this video and complete the exercise(s)

CLICK HERE

REMEMBER:

- For revolving, you need an **axis and a profile/face**
- You are **revolving** your profile **around the axis**
- Make sure the **axis is center of your design**

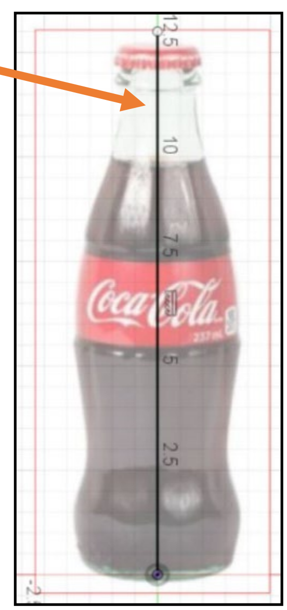
  (Otherwise, the part will be too small or too big)
- **Starting and ending point must be on the axis of revolution**
- After making the cap, **make a new sketch of the bottle** (on the same plane as the canvas) and then revolve

Otherwise follow these steps:

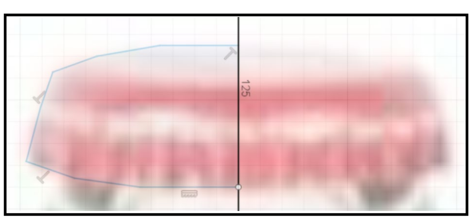
STEPS TO FOLLOW:

1. Click "**Sketch**" and select the **same plane** you put your photo of coke on.

2. Put a line straight down the center (as center as possible) of your photo of coke, this will be your axis of revolution.



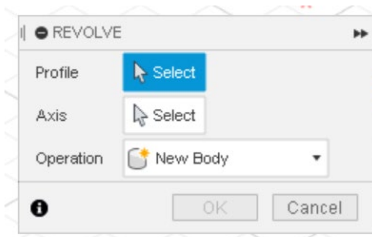
3. Now we are going to sketch the **ONE SIDE of the bottle cap** using the **line tool,** starting from the axis of revolution. **BE SURE TO START AT THE AXIS**



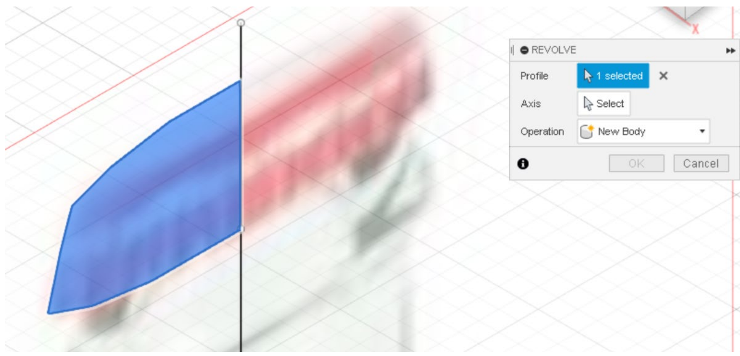
4. Once done, click "**Finish Sketch**"

5. Above create, click your sketch, and then click the "**Revolve**" tool

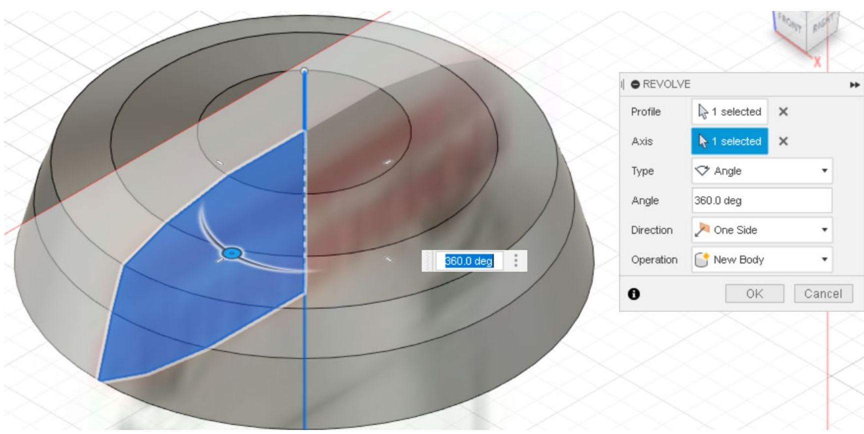
6. On the right you should see this:



7. While on the "**Profile**" tab, click the sketch of the bottle cap



8. Switch to the "**Axis**" tab, click the line down the middle



9. The shape should automatically create and click OK to finalize!

10. You have now created your bottle cap!

Save your work by clicking on the save button at the top left.

Time to create the body! Create a new body/sketch for the bottle.

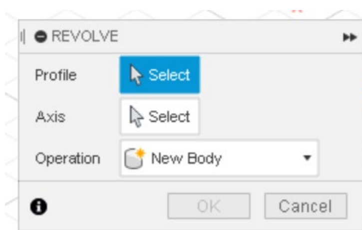
1. Click Sketch and select the **same plane** you put your photo of coke on.
2.
3. Put a line straight down the center (as center as possible) of your photo of coke, this will be your axis of revolution. [(Just like before, click here to see the reference)]()

4. Now we are going to sketch the **ONE SIDE of the bottle cap** using the **line tool,** starting from the axis of revolution. **BE SURE TO START AT THE AXIS**

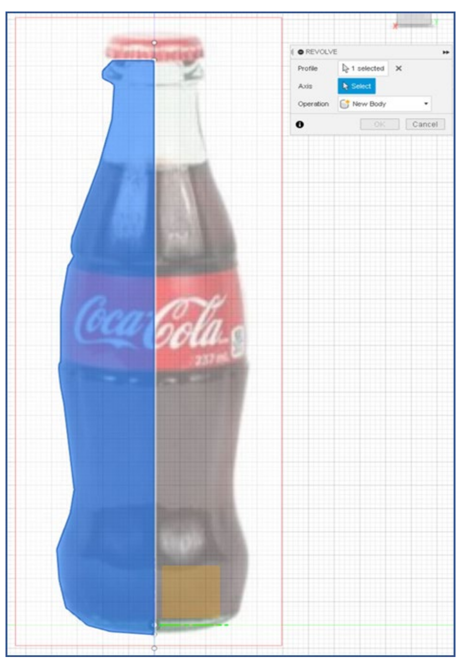
5. Once done, click "Finish sketch"

6. Above create, click your sketch and then click the "**Revolve**" tool

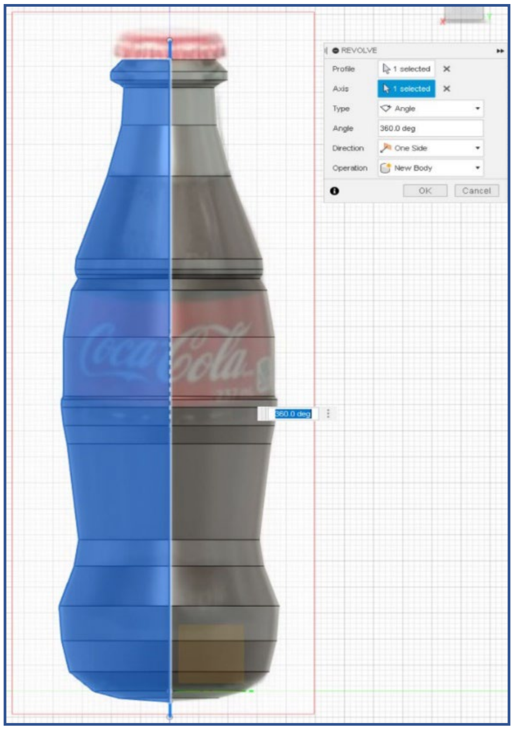
7. On the right side of screen, you should see this:



8. While on the "**Profile**" tab, click the sketch of the body

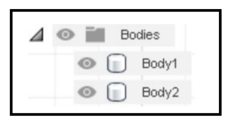


9. Switch to "**Axis**" tab, click the line down the middle, the body will form.



Click **OK** and save with the saving icon at the top left! You will now have a coke bottle!

Make sure the **BOTTLE CAP** and **BOTTLE** are **2 SEPARATE Bodies**.



SPLIT BODY TOOL

We want to split our coke bottle into different sections so we can **add different colours, materials and details to make it look realistic**.

This can happen by **adding different planes to sections** of the bottle

Let us add a plane to divide where the bottle label should go and where the bottle should be filled with liquid.

CLASS VIDEO:

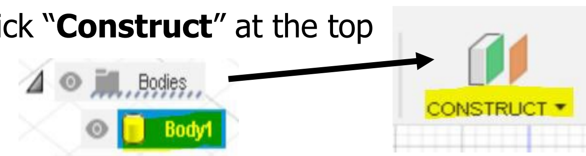
Watch this video and complete the exercise(s)

CLICK HERE

Otherwise follow these steps:

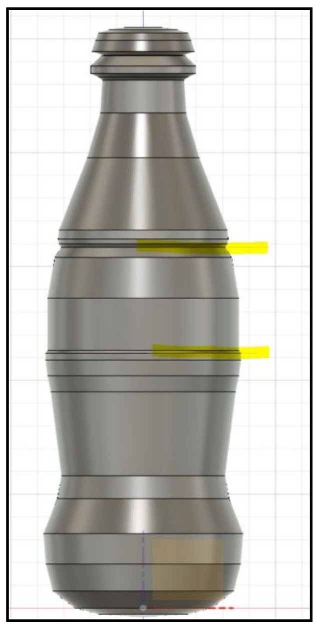
STEPS TO FOLLOW:

1. Click your coke bottle **body** and **then** click "**Construct**" at the top
2. Then click "**Offset Plane**"



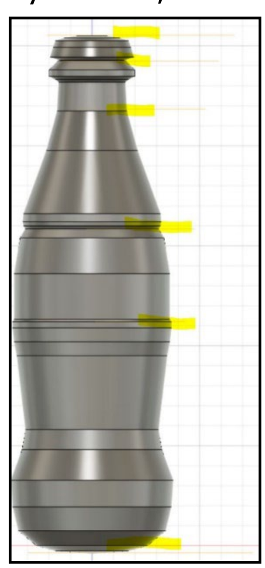
3. We can now move our planes to different parts of the bottles. **To duplicate a plane, copy and paste** (CTRL-C & CTRL-V)

4. First, we will move the bottom plane to the areas we will colour red.



5. You repeat this to have the number of sections you want, in this example there is a section for:

- Red Logo
- Liquid
- Empty
- (Glass looking) Parts

6. Once you have placed all your planes, go under Modify at the top

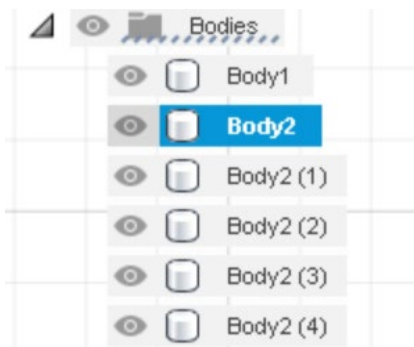
7. Then click "**Split Body**"

8. First choose the body you want to split (the coke bottle)

9. Then switch to the "**Splitting Tool(s)**" tab and click each plane you created, then click OK.

10. You will then notice there are new bodies created under the Bodies tab. This are the new sections you have created!

    Name these new bodies the part, this is for organization!



11. Now you have different sections that you can colour later in the future!

You can imagine how the split body tool can help you create realistic renders of the part you created.

Be sure to try it on the upcoming homework practice!

HOMEWORK AND PRACTICE

After all you have learned, try to make these examples!

Checkmark them off when you have done them.

Exercise 1 ☐

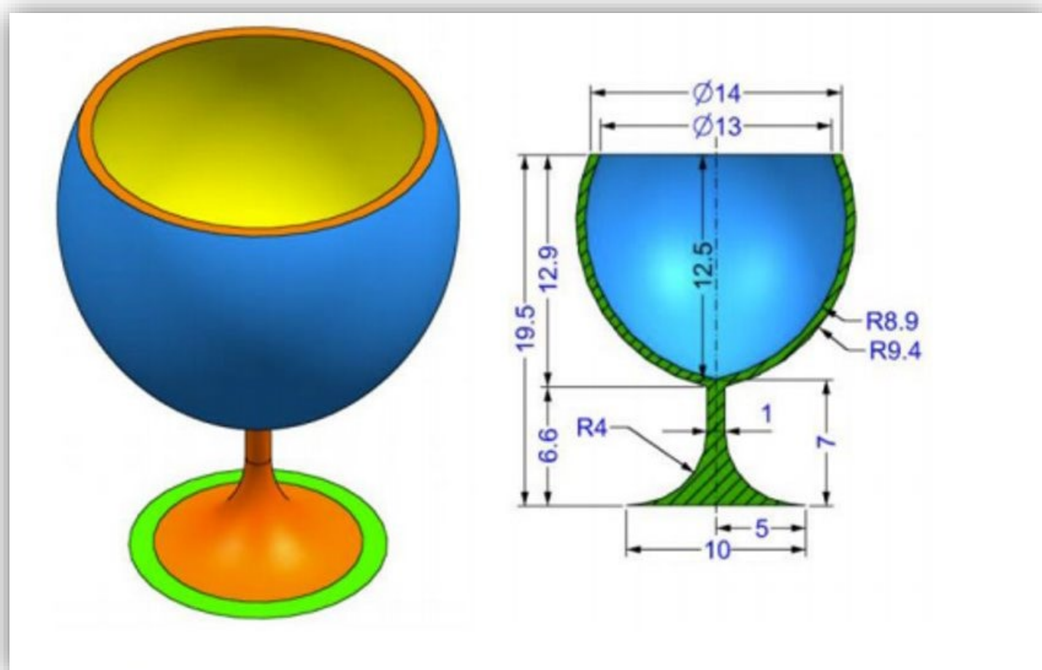
1. Attempt a floor lamp.

- Size it appropriately
- Imagine the section of the lamp
- Think about the dimensions will you put for the floor lamp

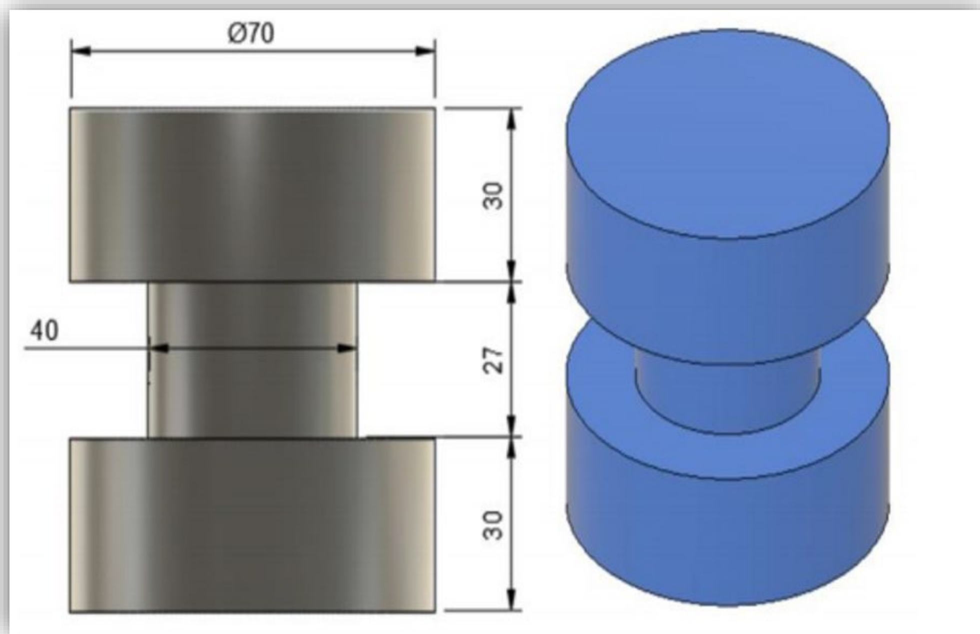
When you attempt these designs, think about how you can apply**:**

The Revolve tool and Split tool

Exercise 2 ☐



Exercise 3 ☐

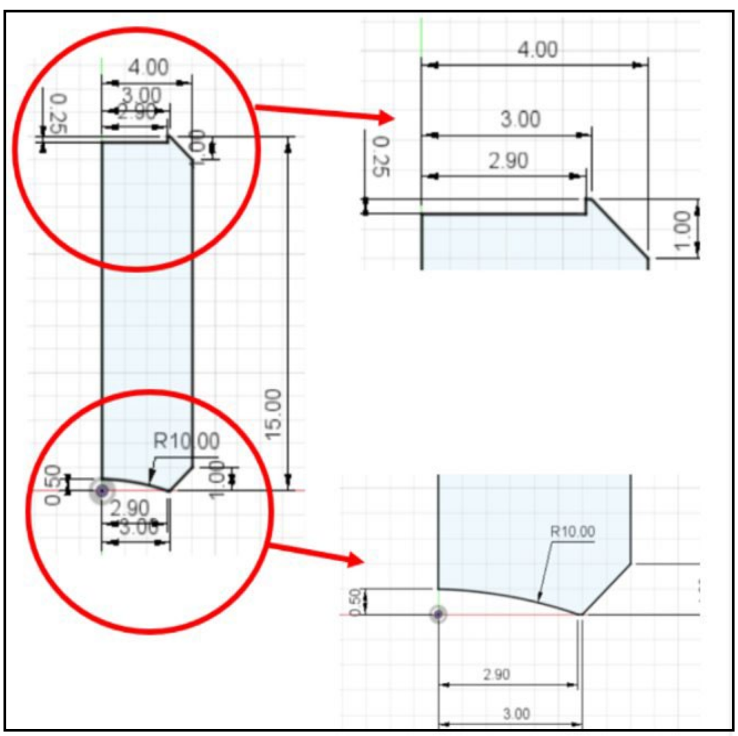


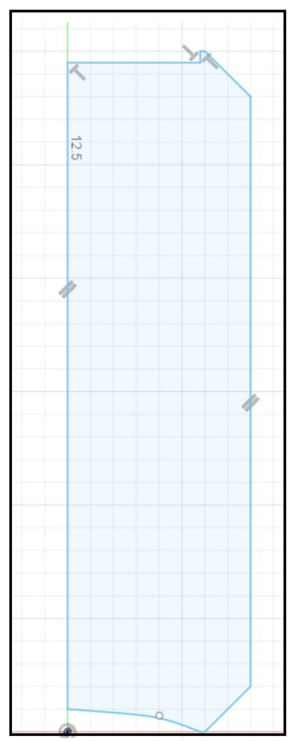
5: SHELL AND RENDERING WITH REVOLVE

Before we start, you must create another coke bottle.

Create this coke bottle:

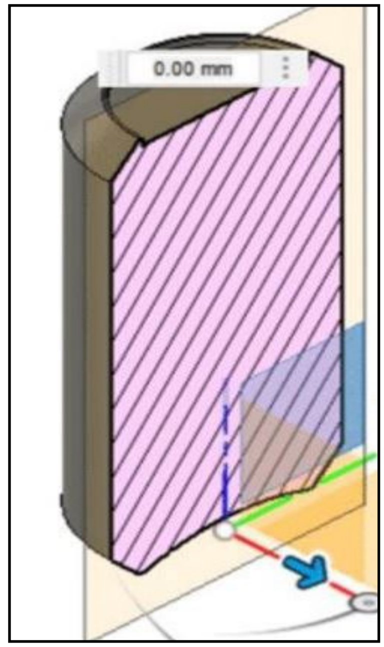
Use these dimensions and sketch:





SOLID VS HOLLOW BODIES

When we use a section/cut view to see the inside of the part, we see that the can is filled with material, as in it is solid inside. Therefore, for this example, <u>should the inside of the coke can be solid or hollow?</u>



Are they hollow or solid?

The inside of a coke can is hollow

So how do we make the hollow?

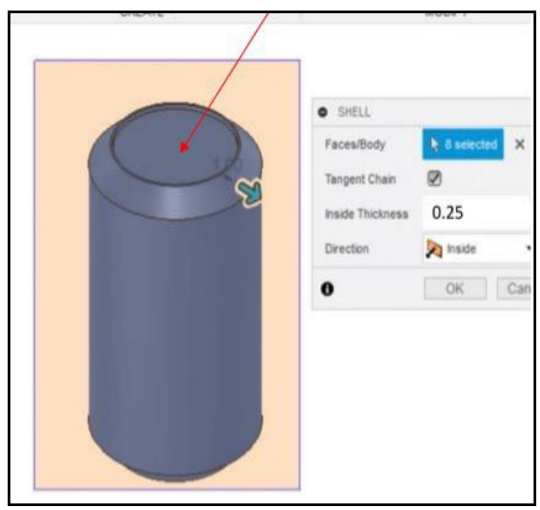
We use the SHELL feature in Fusion 360

SHELL FEATURE

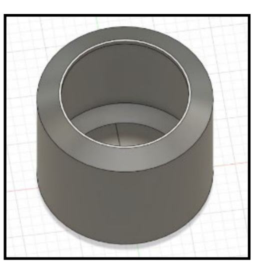
The shell tool can be found in the <u>Modify</u> menu. It will be used to make your part hollow on the inside

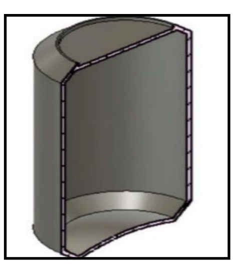
Now, shell your coke can!

1. Click Modify and then click "**Shell**"
2. For Faces/Body select all the faces of the part
   You **can click the body** under "**Bodies**"
3. Next specify the inside thickness as 0.25mm
4. Click **OK**



When you are done the inside should look like this:
(You can check by deleting the top face and then undo the delete after you have checked.)





INTRODUCTION TO RENDERING

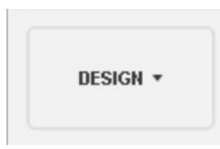
Rendering is Fusion360 allows you to **add material, colour and lighting to your model to make it look more realistic and aesthetic**.

For this example, we can create a rendering of the coke can!

Use this design that we created HERE

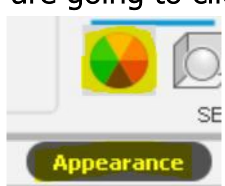
RENDERING

1. At the top left, click "**Design**"

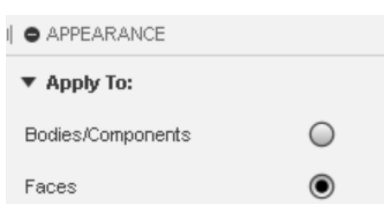


2. Then switch it to "**Render**", this will take you to the Rendering space in Fusion 360.

3. To colour our coke bottle, we are going to click "**Appearance**"



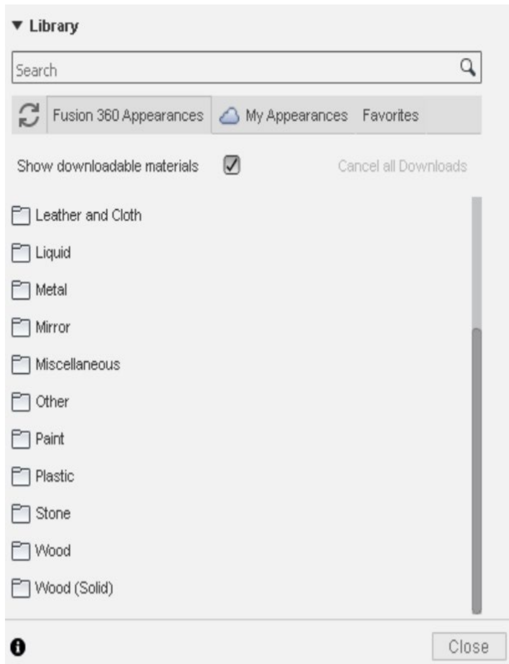
4. It gives us an option to select and colour either "Faces" or "Bodies/Components", let us select the "**Faces**"



5. There are **different materials** and paint colours to change different bodies, **look around** and decide what types you want to use.

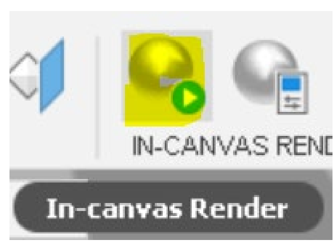
   Search and change the appearance of parts through here:

Once you find a colour or material that you like, drag it into "**In This Design**" and then apply it to your part.

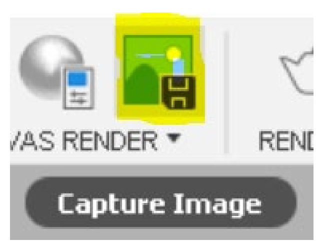


Or you can apply the colors by **dragging the colors to the faces of the part**

6. Once you have created your desired design click the "**In-canvas Render**"



7. It might look a little pixelated but that is just the materials showing, you can capture the look of your coke bottle and show off what you have made! Click "**Capture Image**" to take a photo of what you have rendered

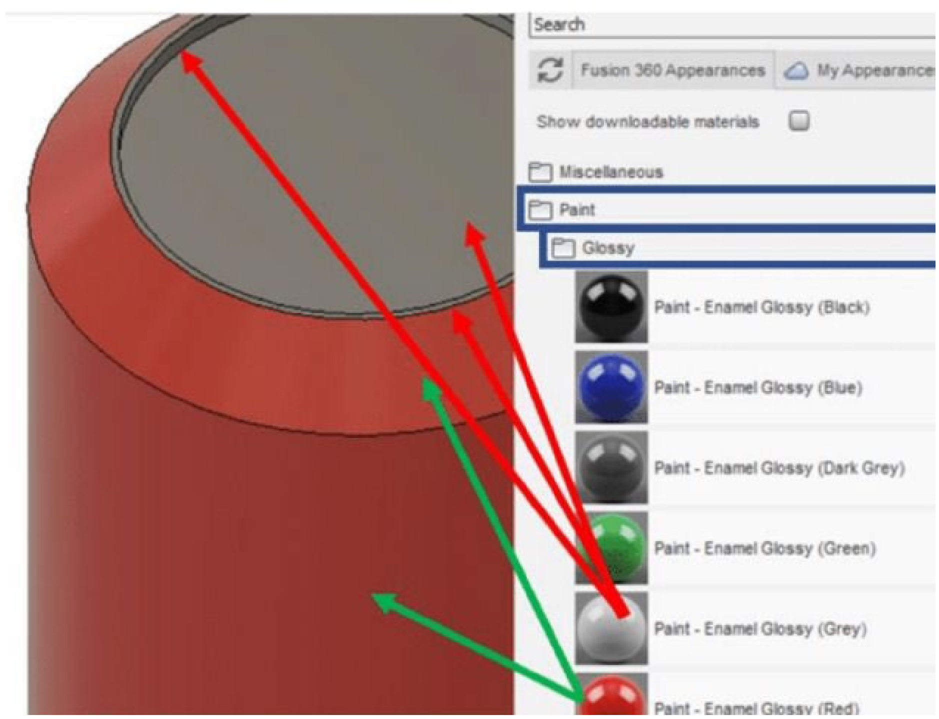


HOMEWORK AND PRACTICE

The homework for this lesson is simple! All you need to do is **COLOUR AND RENDER** the part you created HERE

Try to make this coke can: look like this:

Trying using rendering materials that are shown below:

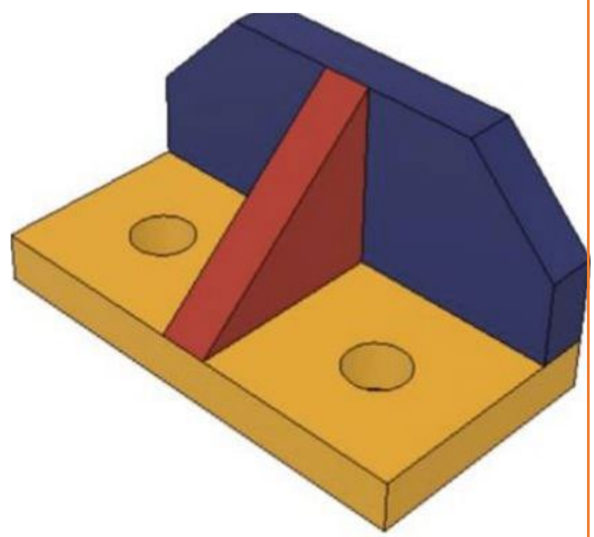


6: MULTI-SKETCH BODY

The Multi-Sketch Body is the concept of creating **multiple sketches on one body to create complex a part**. You can see this through today's challenge.

Today's challenge is to create this part

- This part is more complex than any part we have modelled before.
- However, we can break it into **3 smaller and simpler bodies that we know how to model**.
- First, we have the base (in yellow) with 2 circular holes
- Next, we have a wall (**in dark blue**) with 2 triangular shapes cut out
- Finally, we have a bridge (**in red**) shaped like a triangle

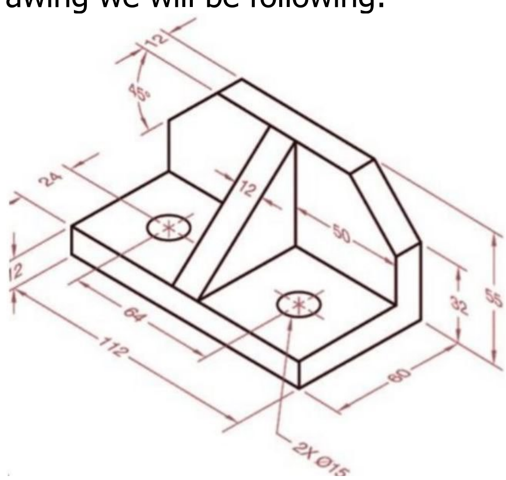


Let's try to create this shape!

SKETCHING ON THE BODY

HOW TO DO THIS

This is the orthographic drawing we will be following:



We will start with the yellow base

Question: Which plane should I choose when creating the sketch?

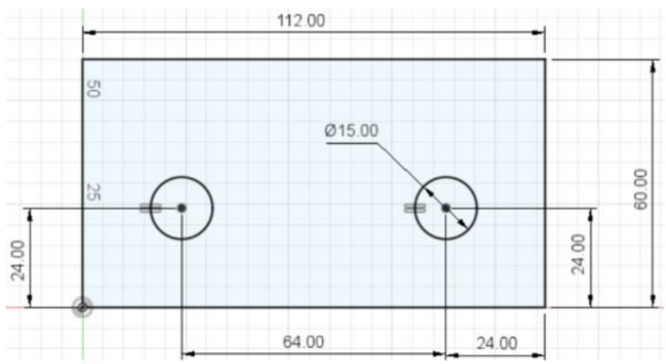
OPTION A: Front plane

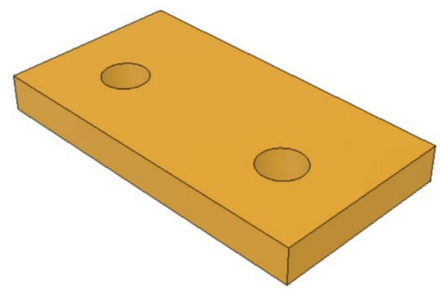
OPTION B: Right plane

OPTION C: Bottom plane

Correct answer is OPTION C. Create a sketch in the bottom plane.

THE BASE (YELLOW)

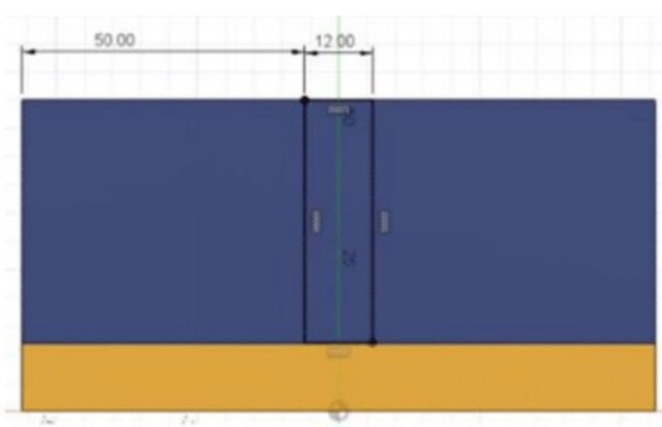


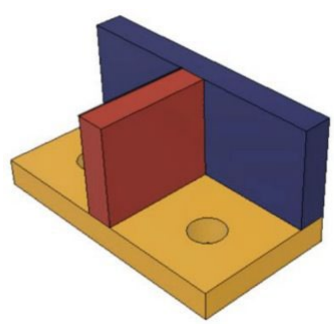


Create the base of this object and extrude it by 12mm

THE WALL (BLUE AND RED)

1. Click on "**Create a sketch**"
2. Click on the <u>top of the base</u> you just created for the plane

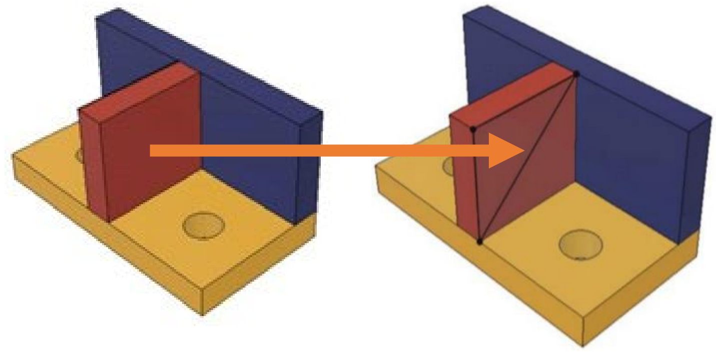




Draw the bridge using a "2-point rectangle" as shown in the figure. Extrude the rectangle by <u>48mm</u>.

THE WALL (BLUE AND RED)

Now, we want to cut out the triangle.



1. Click on "**Create a sketch**"
2. Click the side of the red square
3. Draw the **triangular shape we want to cut out**
4. Finish Sketch
5. Extrude / Cut out the triangle

INTRODUCTION TO CHAMFERING

CLASS VIDEO:

Watch this video and complete the exercise(s)

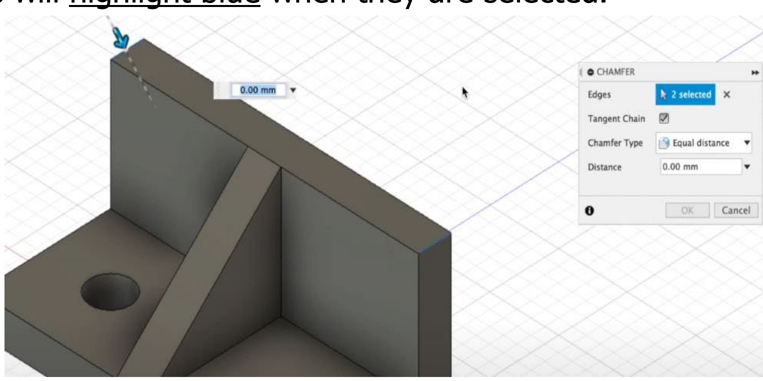
CLICK HERE

Otherwise follow these steps:

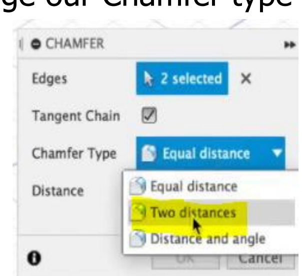
STEPS TO FOLLOW:

1. Find "**Chamfer**" under Modify

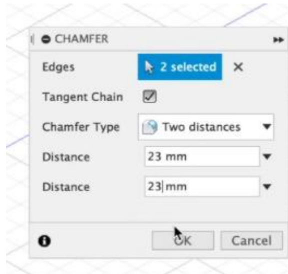
2. You are now prompted to select an edge; select the **2 edges you want chamfer**. The edges will highlight blue when they are selected.



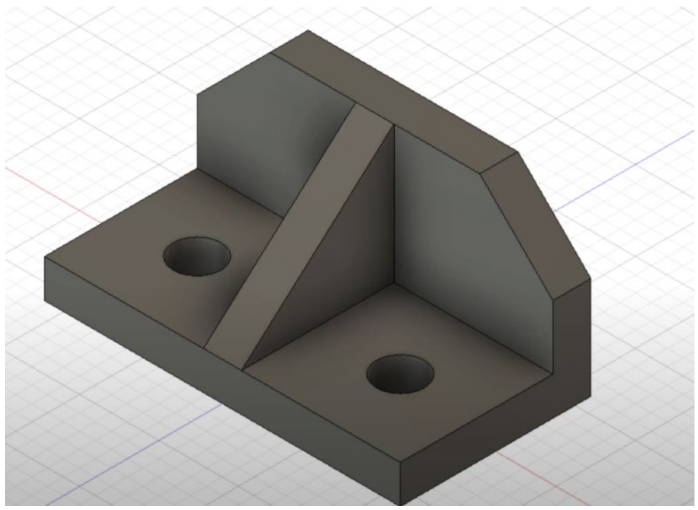
3. We are now going to change our Chamfer type to "**Two Distances**"



4. Once you have changed it, enter 23mm for both distances.



5. Click "OK" and you are finished!



You have now finished the design!

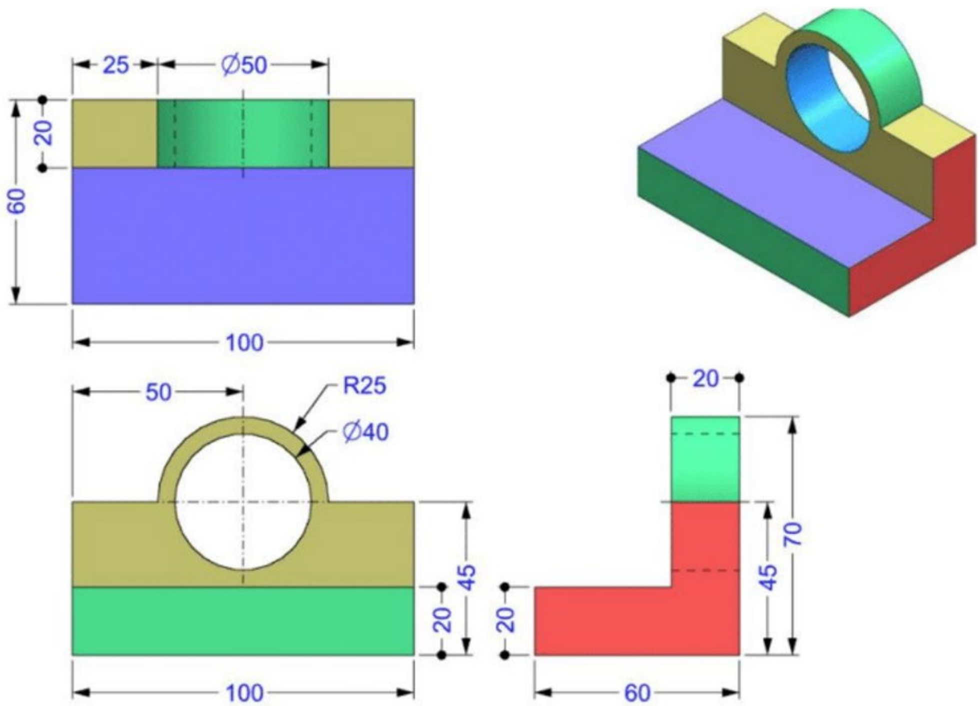
Using the methods of creating sketches on faces, try to attempt all the homework questions below!

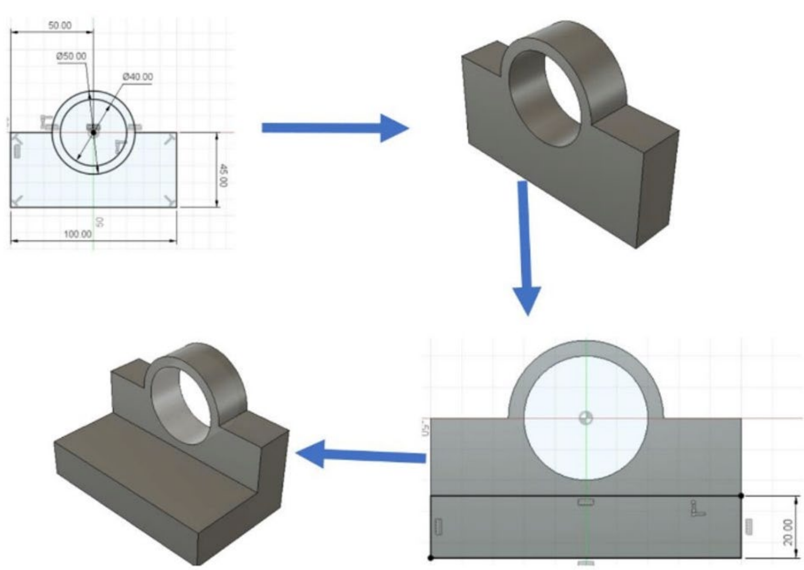
HOMEWORK AND PRACTICE

Follow the designs and try to recreate these shapes using all the skills you have learned in the whole coursebook.

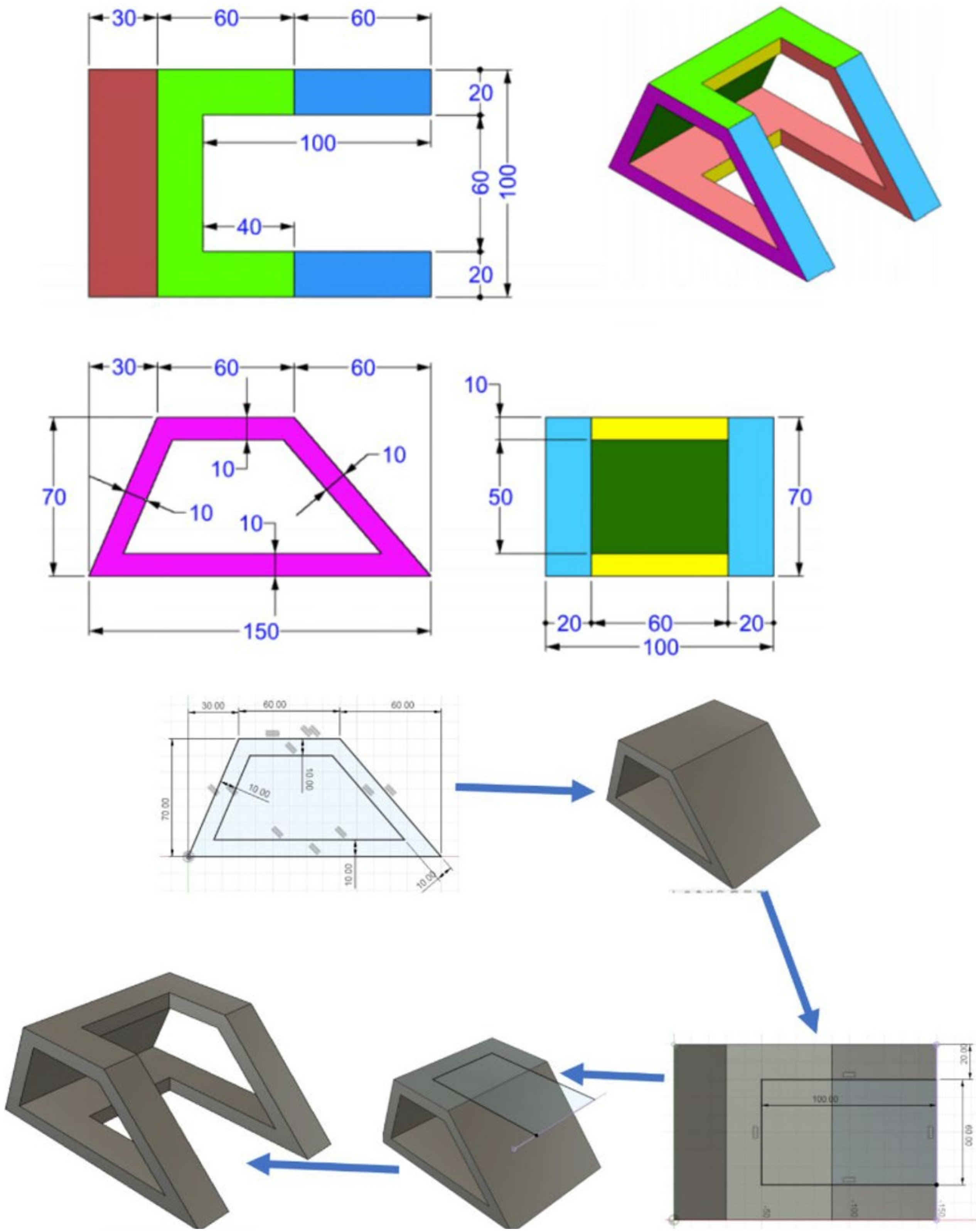
<u>Checkmark them off when you have done them.</u>

Exercise 1: ☐





Exercise 2: ☐



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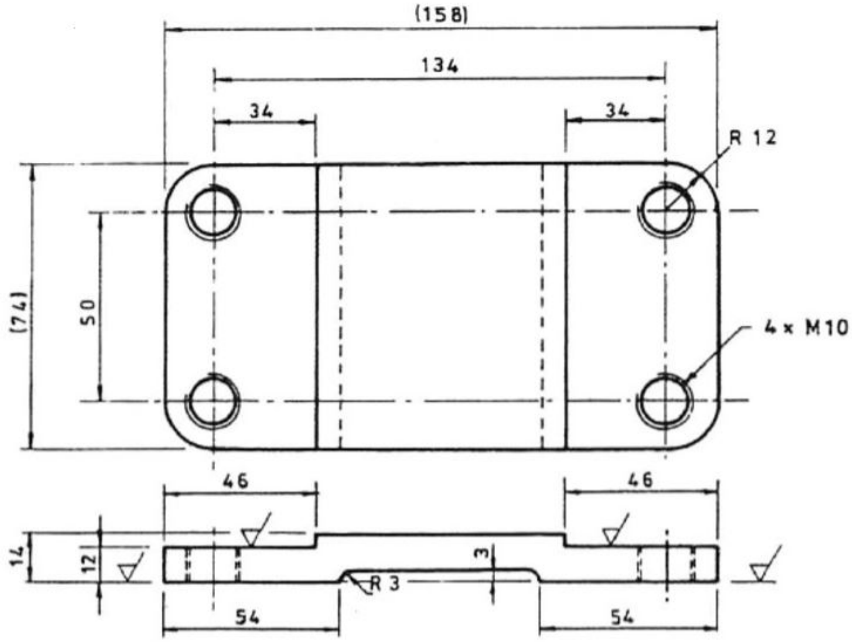
7: INTERMEDIATE DRAWING

The goal for this class is apply all you have learned in these past lessons to this drawing



Refer the [HERE](#) to read this drawing!

DESIGN IT

CLASS VIDEO:

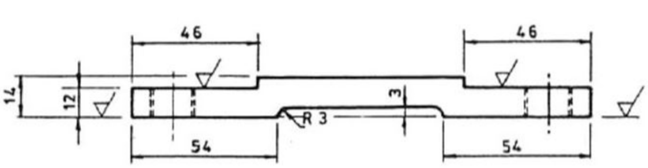
Watch this video and complete the exercise(s)

[CLICK HERE](#)

Otherwise follow these steps:

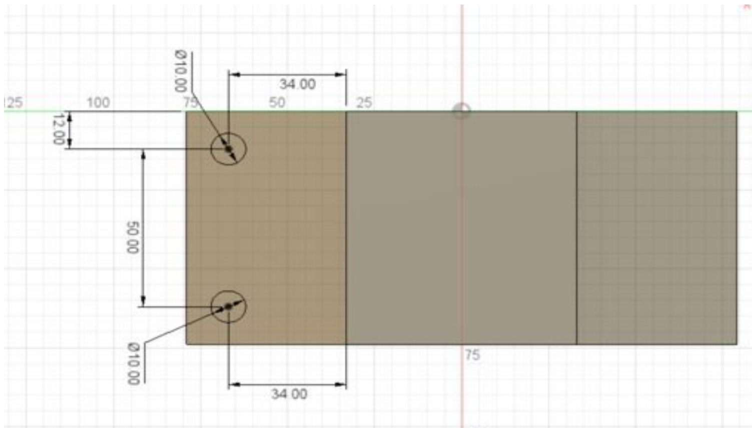
STEPS TO FOLLOW:

1. Draw the sketch of the side view with the correct dimensions.

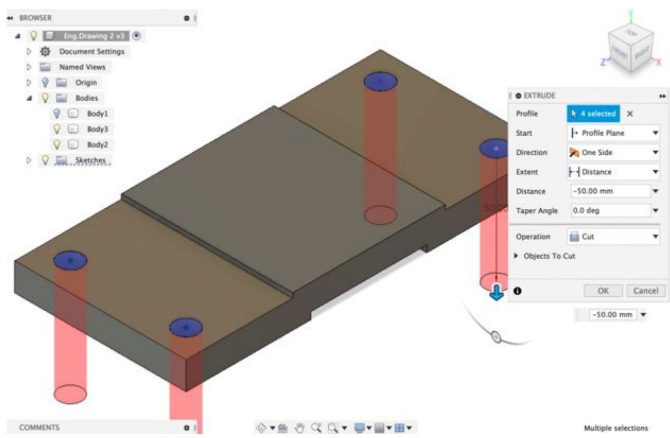


2. Extrude the sketch 74mm (sideways)

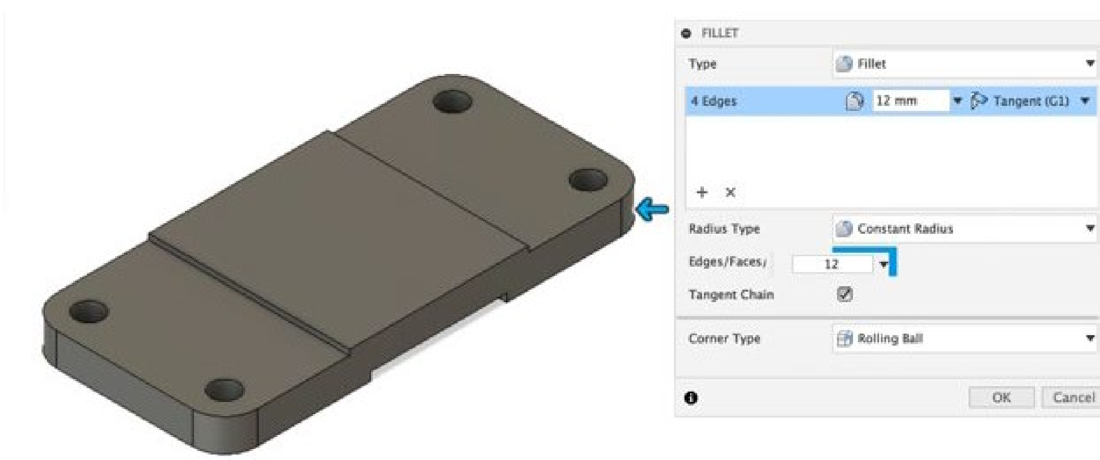
3. Create two circles with the correct dimensions. Do the same on the other side.



4. **Extrude cut** the circles all the way through



5. **Fillet** the 4 edges (12 mm radius)

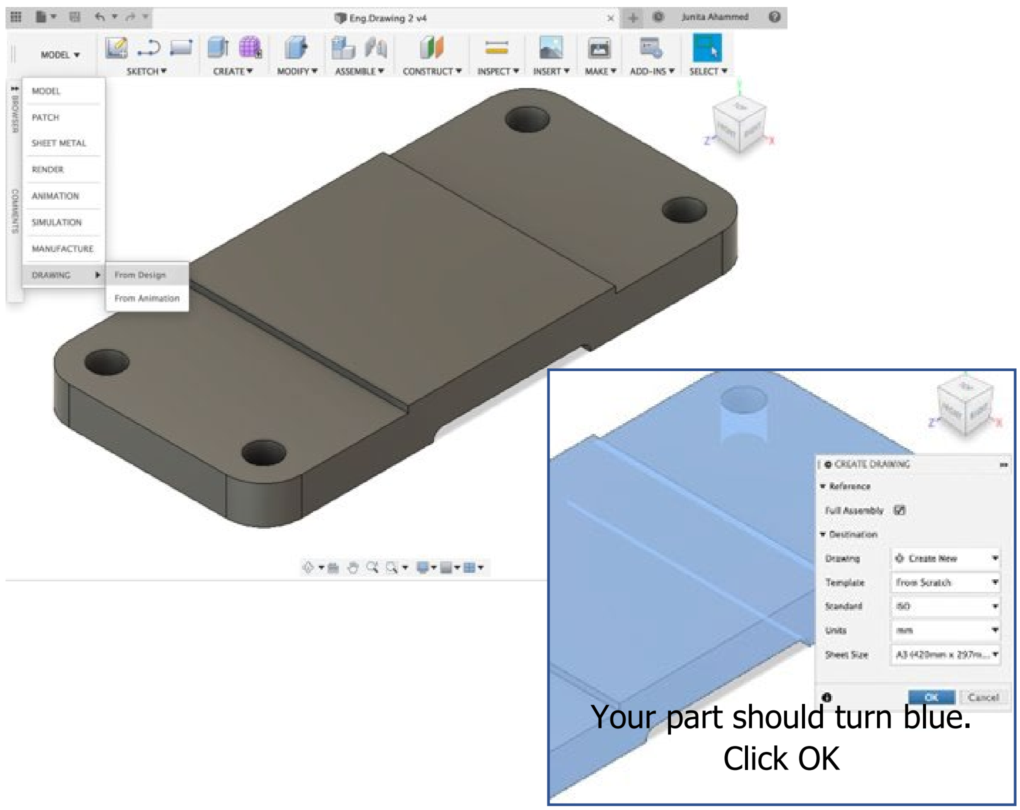


You have now finished the design!

MAKING AN ENGINEERING DRAWING

We are now going to make an engineering drawing for the part we have just created. These are almost like **blueprints and dimensions**. Refer to HERE to remember what an engineering drawing is.

1. Under model, find drawing and click "**From Design**".

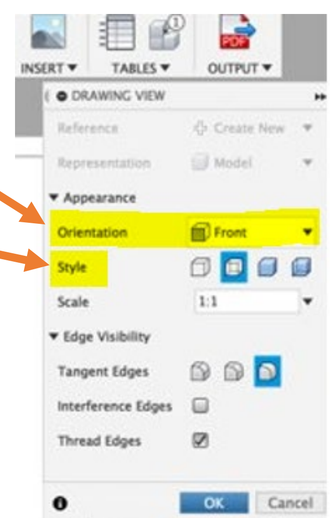


2. A box called "Drawing view" should appear.

3. Make sure beside "Orientation" it says "**front**."

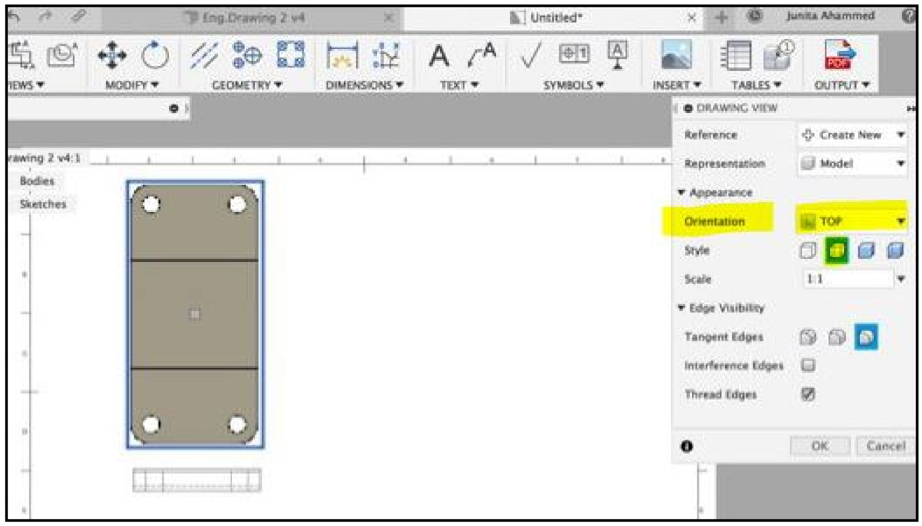
4. Beside "Style" choose "**visible and hidden changes**".
   - Visible and Hidden Changes = the second cube

5. Click OK.

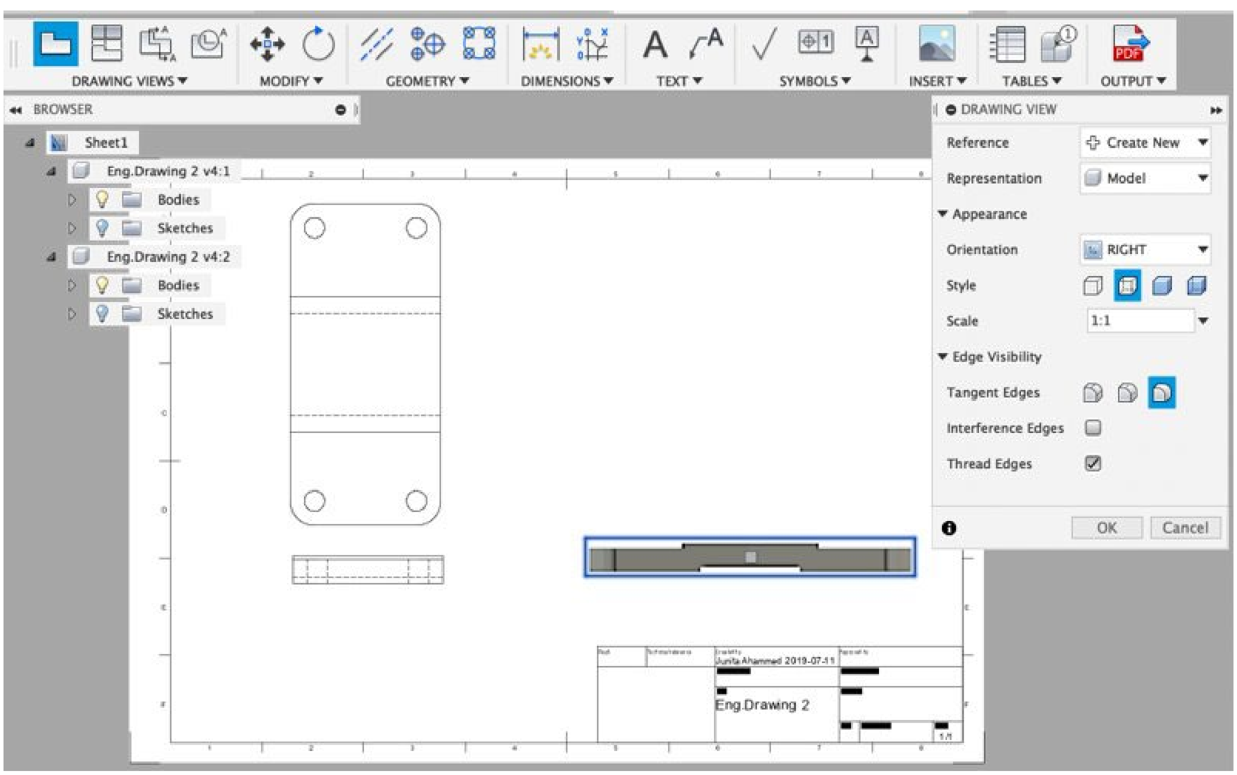


CREATING MORE VIEWS:

1. Click "**Base View**" in "**Drawing Views**" and this time choose TOP for orientation.

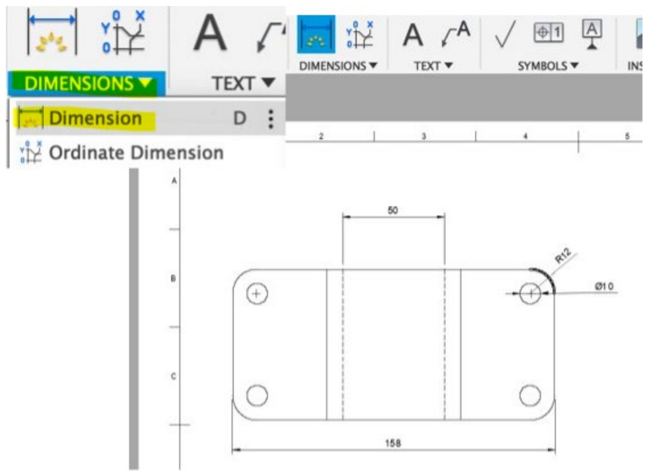


6. Continue creating more views by changing the orientation!



7. Click and drag on the view if you want to move it around.

8. Go to “Dimensions” and click “**Dimension**”.
   Click on **different lines** on your views to **create dimensions**.



<u>Try to add as many dimensions as you can to your views!</u>

When you are done you can save a copy of your engineering drawing.

You can now apply this to any design you want to share with peers, co-workers, or teachers. This will help the people viewing your design to know the dimensions and details of the shape!

FINAL NOTES

You have now completed this workbook! Congratulations.

You have now learned all the content in MEC07. You can apply this to designs you want to try and further courses at Exceed Robotics! Be sure to use all the resources that this coursebook provides.