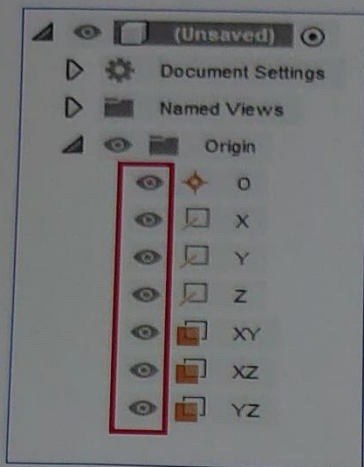




## The Cube



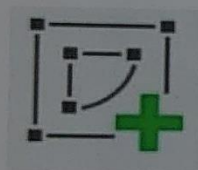
Step 1 – creating the 100mm x W= 100mm x H= 100mm cube



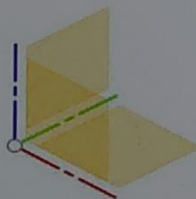
If not visible turn on your X, Y, Z planes by toggling the Eye symbol (See image)

Planes can be turned on or off for clarity

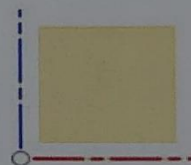
Select sketch icon



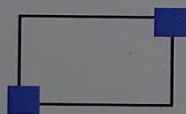
Select (top) plane



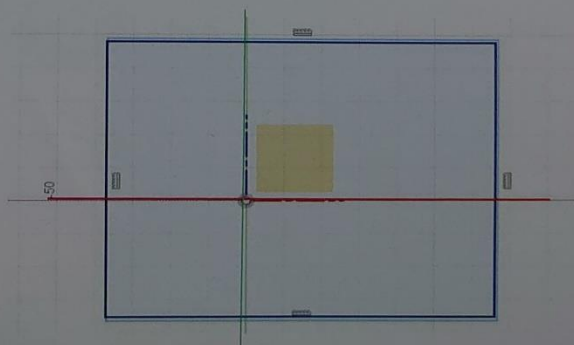
This is the sketch window



Select Rectangle icon



Sketch out a square that covers all 4 sections of the grid



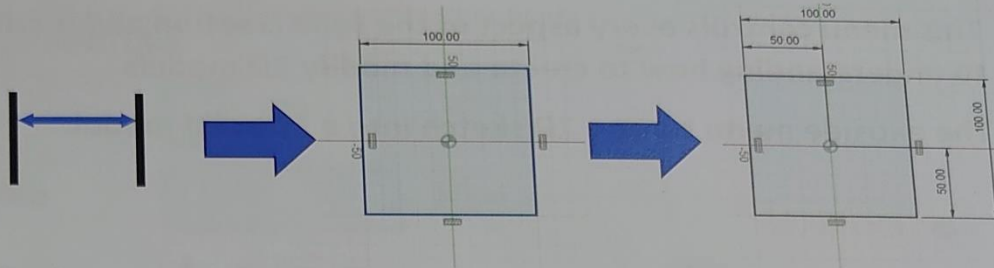
## 3D 360 CAD Training course worksheet exercises



Select Dimension icon  
or use 'D' key

Add dimensions and  
modify to make a square

Completely dimension  
as shown



Dimension the square as shown below.

Make sure you do not add too many dimensions as you will over-constrain your sketch.

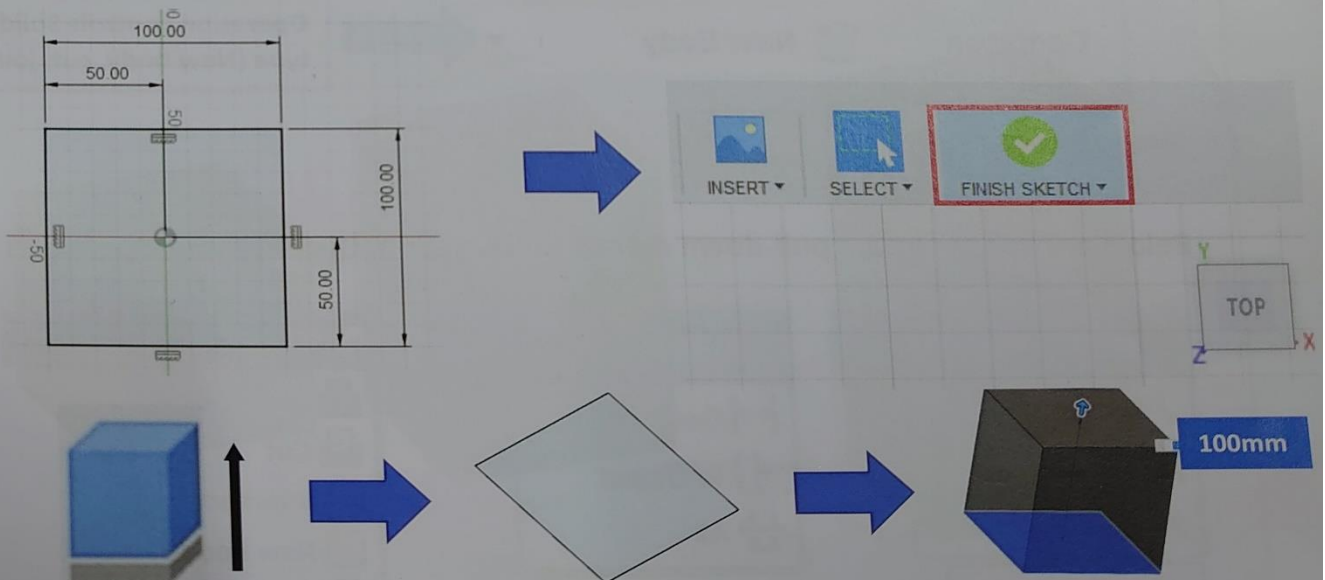
If the Fusion 360 over-constrain menu comes up just choose to **Cancel**

### Fusion 360

Adding this dimension will over-constrain the sketch. Choose OK to create a Driven Dimension.



Once sketch is correctly dimensioned select the "FINISH SKETCH" Icon in the top right



Select the extrude icon

Select your 100x 100mm sketched profile.  
**Note -** If you cannot see your square, it is probably hidden and needs turning on in the tree. Just like we did with the planes

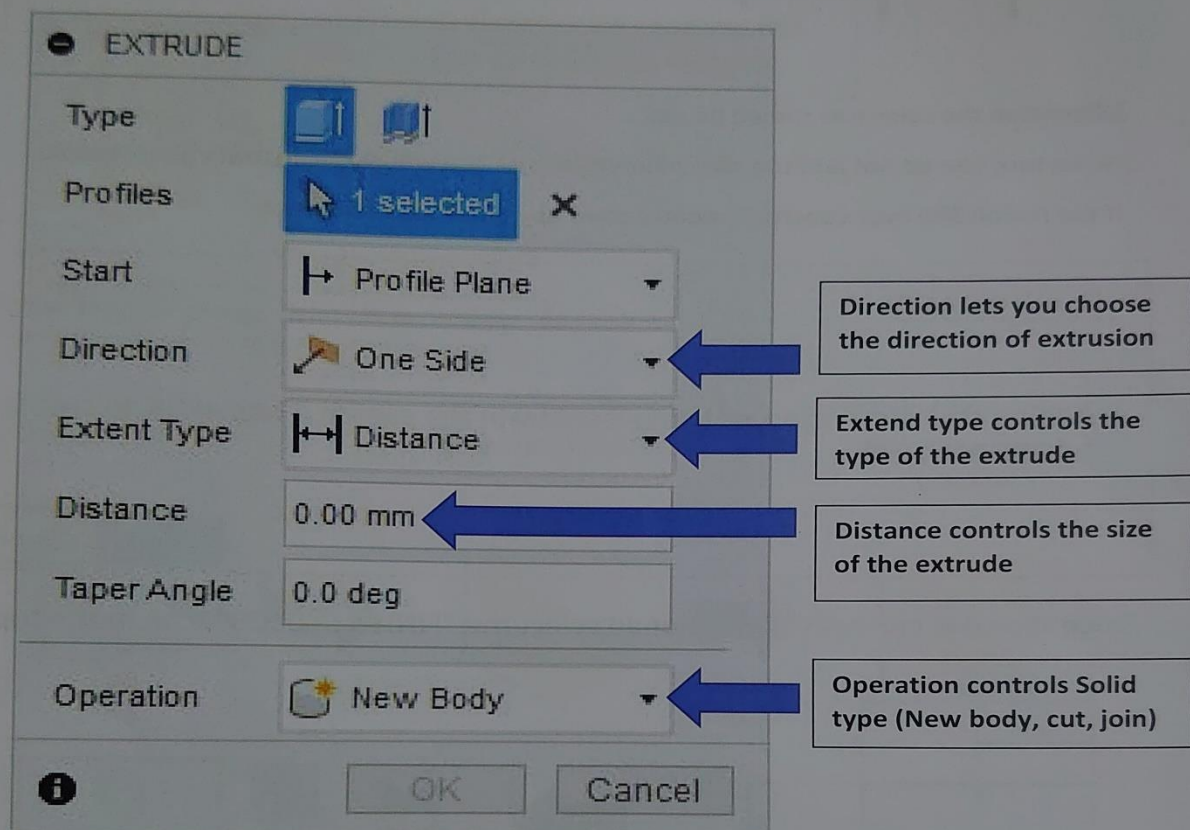
To create your 100mm Cube  
Select the 100 x 100 square by  
typing in a 100mm value



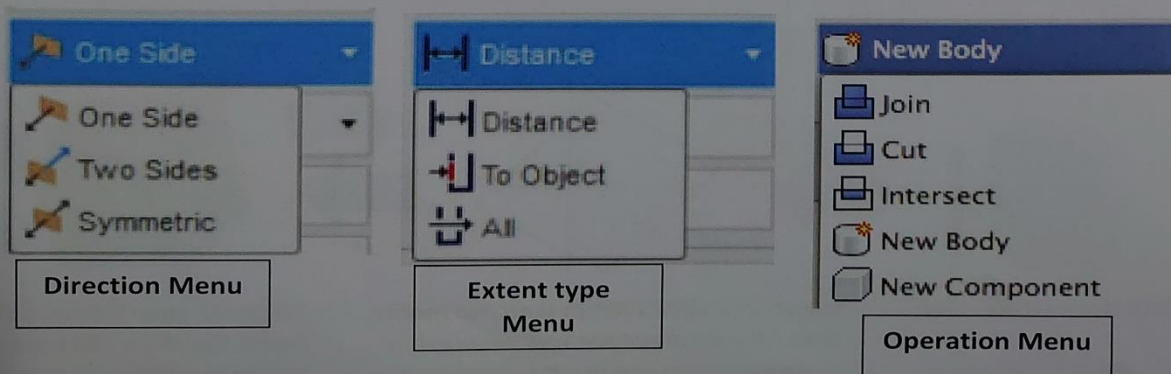
Selecting the Extrude icon brings up the Extrude menu.

This menu controls every aspect of the solid creation and is critical to understanding how to create and modify 3D models

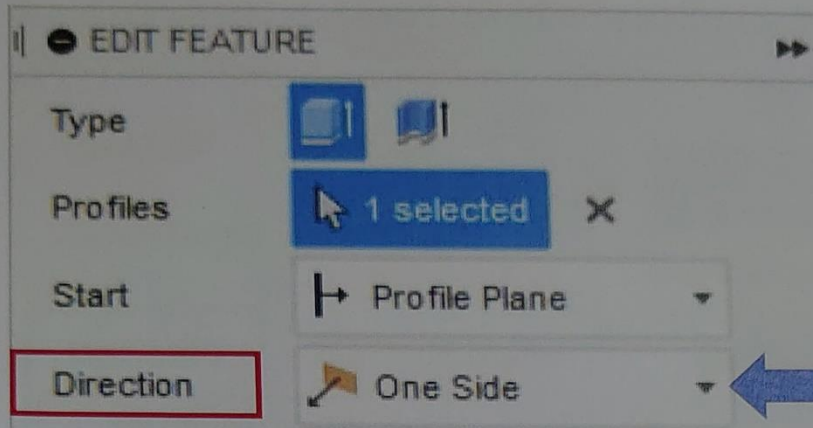
The extrude menu turns a 2D sketch into a 3D solid model.



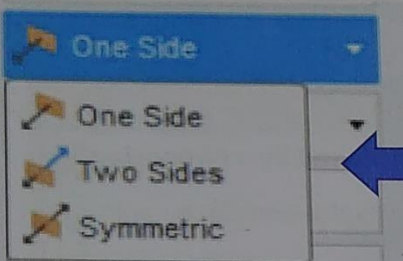
Below are images of the pull-down menus for Direction, Distance and Operation



## Direction Command



Direction lets you choose the direction of extrusion



Direction Menu lets you choose

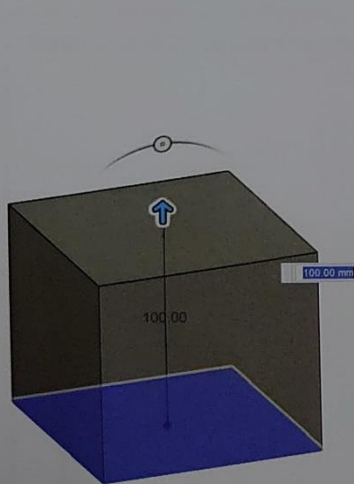
**One Side** – Only extrudes on one side of the sketch

**Two Sides** – Extrudes on two sides.

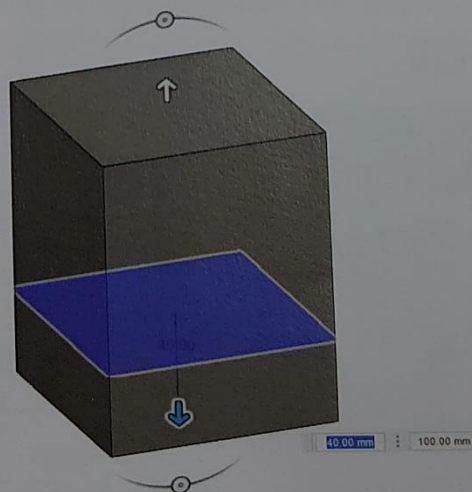
Each side can be a different length

**Symmetric** – Extrudes equally on both sides of the sketch

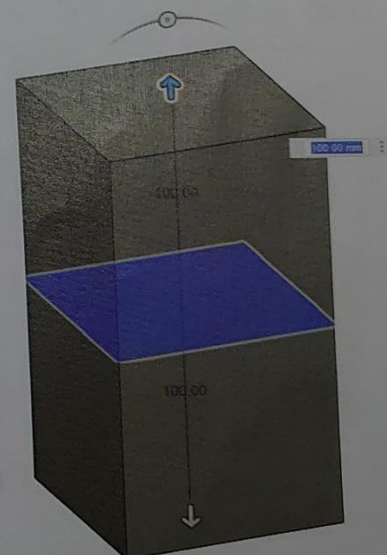
Below are 3 images showing examples of the 3 extrusions variants



One side

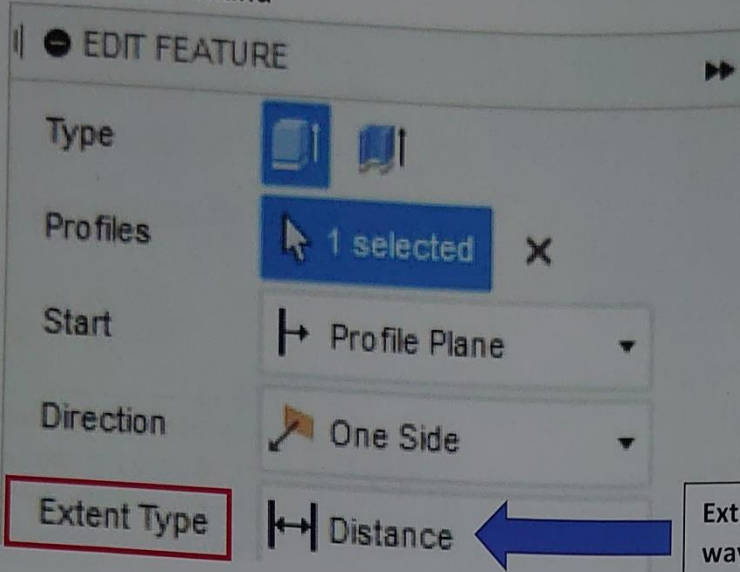


Two sides

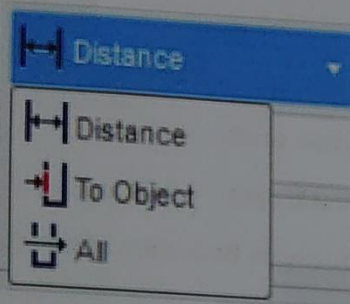


Symmetric

### Distance Command

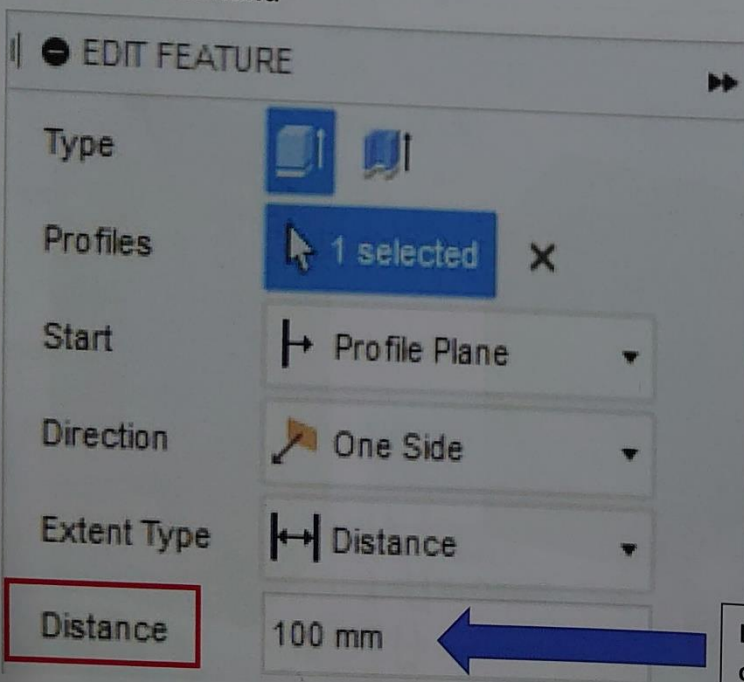


Extent Type controls the way an extrude is projected



**Distance** – Solid size is controlled by a dimension  
**To object** – Cut or extrude to the next face  
**All** – Cuts or extrudes through everything

### Distance Command



Distance controls the size of the extrude



Changing the value of the Distance dimension sets the size of the extrusion of the model

## Operation Menu

Distance	100 mm
Taper Angle	0.0 deg
Operation	New Body
	OK Cancel

Operation lets you choose from New Body, Join, Cut & Intersect

New Body

- Join
- Cut
- Intersect
- New Body
- New Component

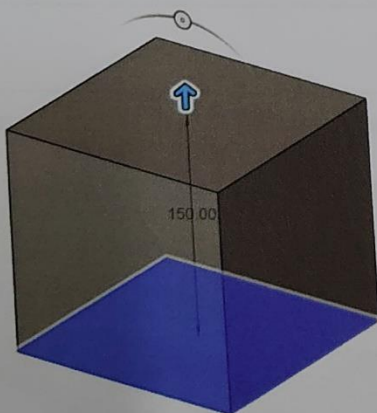
Operation Menu lets you choose the following options

**New Body** – Used to create the first solid feature

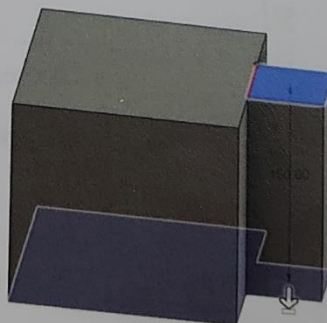
**Join** – Allows solids to be added to other solids

**Cut** – Removes material from other solids

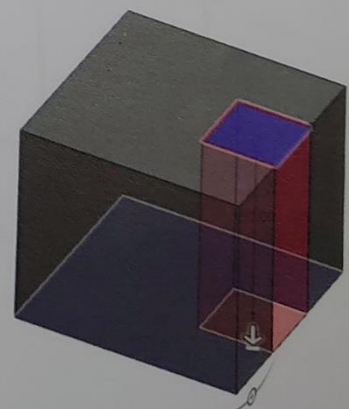
Below are 3 images showing examples of the 3 extrusion types (New Body, Join & Cut)



New body



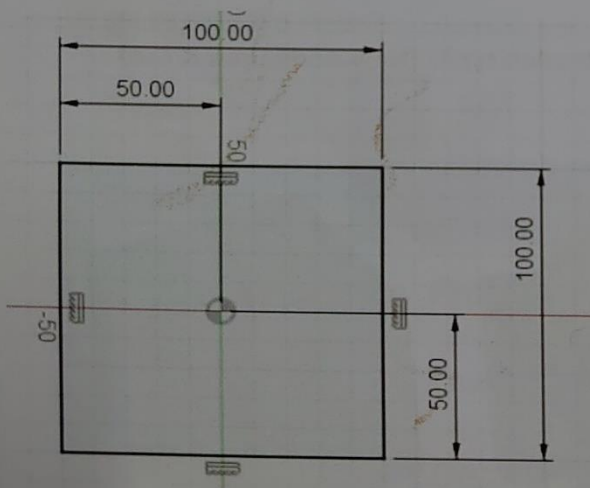
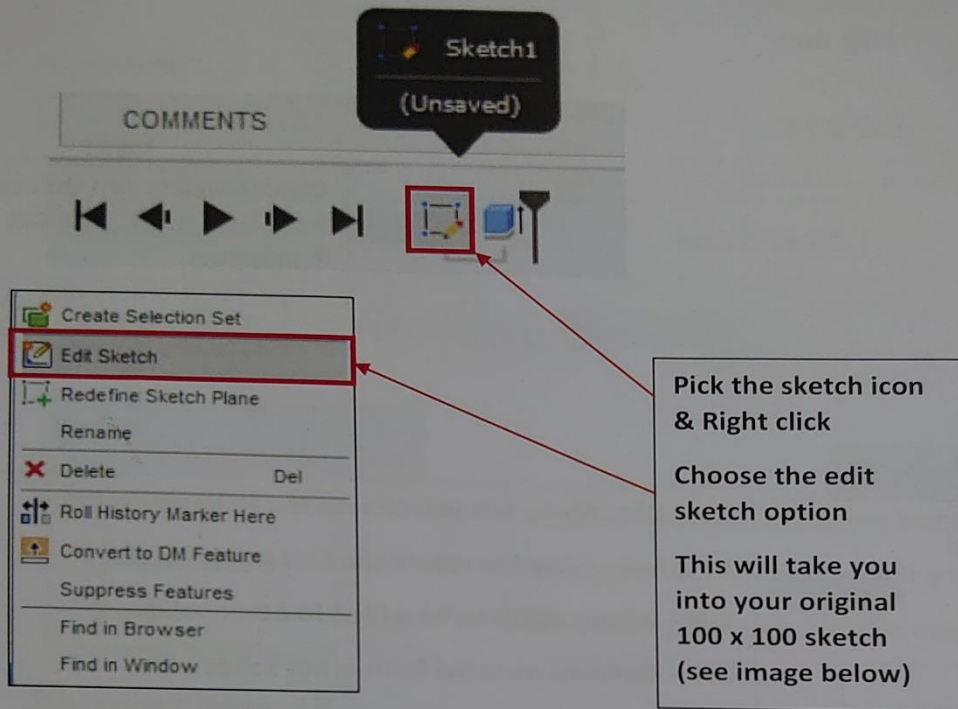
Join



Cut

## Step 2 – Modify your 100mm Cube to a D= 200 x W= 150 x H= 20mm Cuboid

At the bottom left of the screen you will see the model tree.  
This can be found in different locations depending on the CAD package



To modify the size of the cube  
Double Click on the dimension and change each one to the new sizes

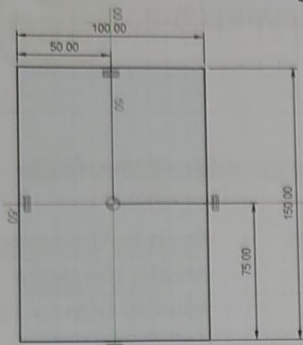
To create the new 200 x 150 x 20mm cube  
You must change the following dimensions

- 1- 100 to 150mm
- 2- 50 to 75mm

Modify the sketch dimensions to replicate the sketch as shown below

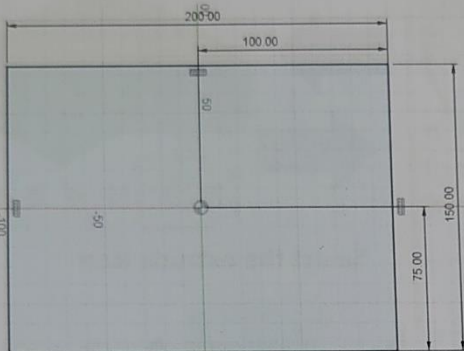


# 3D 360 CAD Training course worksheet exercises



To complete your new 200 x 150 sketch  
You must change the last 2 dimensions

- 3- 100 to 200mm
- 4- 50 to 100mm



When all dimensions are correct select the Green Tick to exit sketch

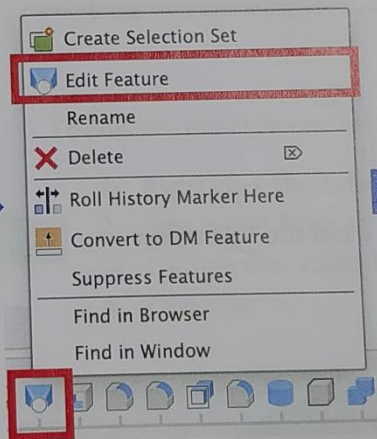
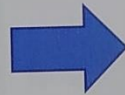


Your model is now the correct shape but not the correct height.  
Reduce the height from 100mm to 20mm

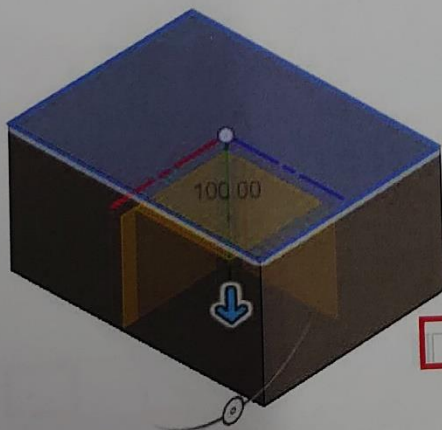
Right click the  
Extrude feature from  
the model tree

Choose the Edit  
Feature option

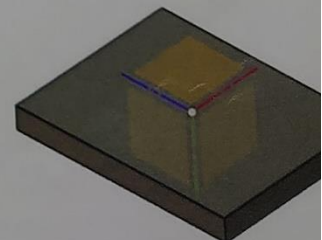
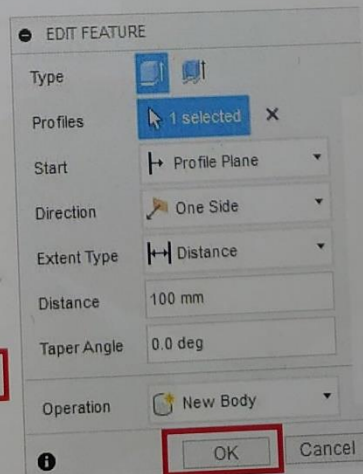
This will take you  
into the EDIT  
FEATURE Menu



Change 100mm  
Dim to 20mm  
Then select OK



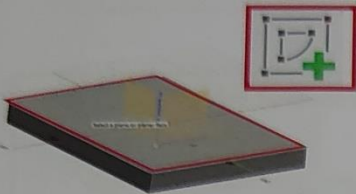
100 mm



200x150x20mm

### Step 3 – Add 120mm diameter, 50mm high cylinder on top of the Cube

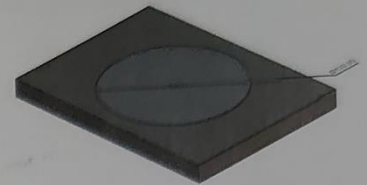
Select the top face of your solid.  
Then select the sketch icon



Select the circle icon from  
the menu tool



Snap to the center & drag out to  
create the circle  
Modify Dim to 120mm



When all dimensions are correct select the Green  
Tick to exit sketch



INSERT ▾

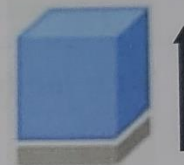


SELECT ▾



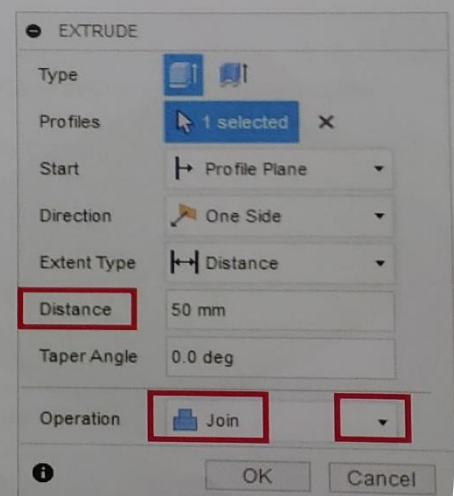
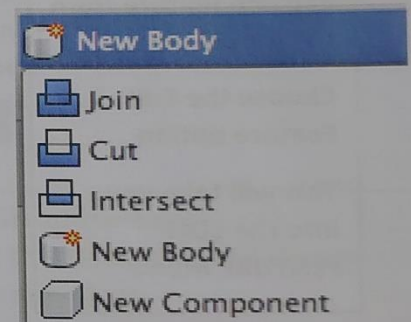
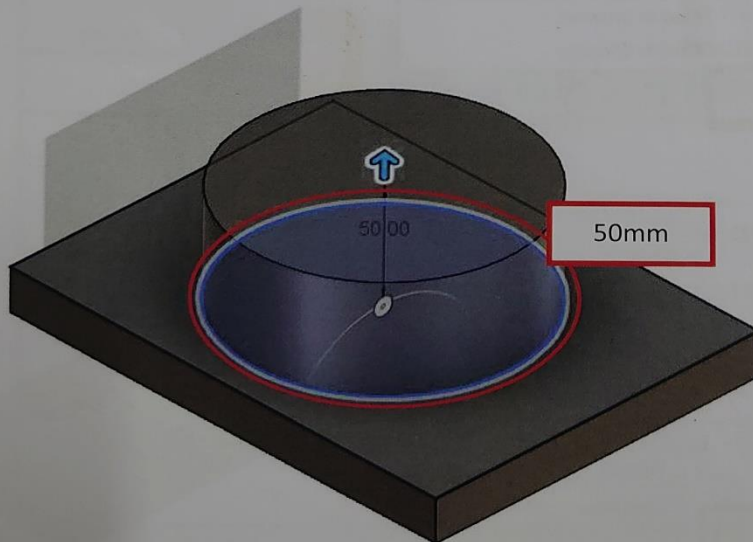
FINISH SKETCH ▾

Select the extrude icon



To Create the cylinder first

- 1- Select the circle sketch
- 2- Set distance to 50mm
- 3- Select the **Join** option from the Pull down

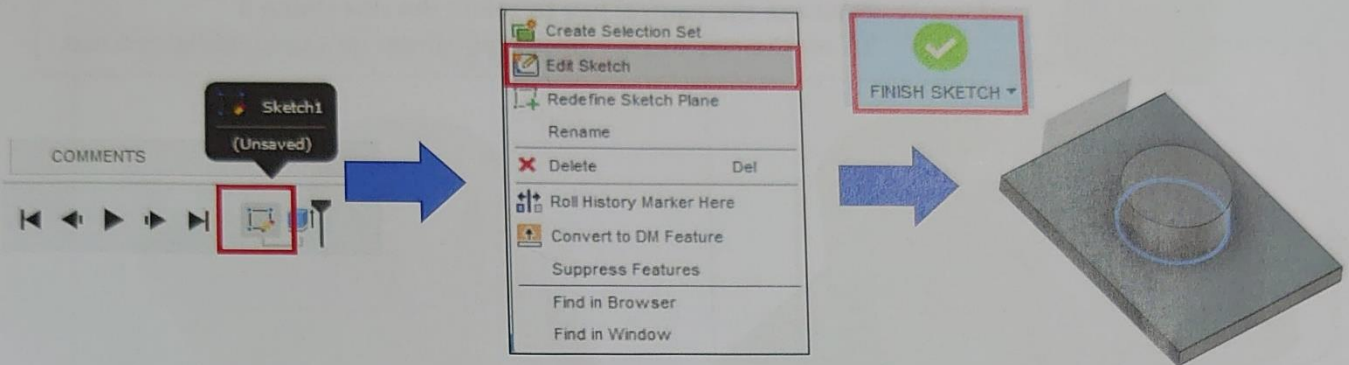


Step 4 – Reduce the diameter to 80mm and increase the height to 100mm

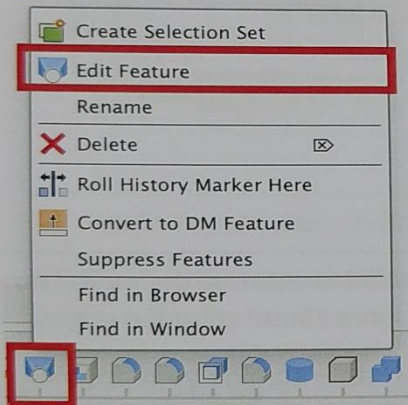
Select the 120 Dia sketch icon

Edit sketch dim from 120mm to 80mm

Edit sketch dim from 120mm to 80mm

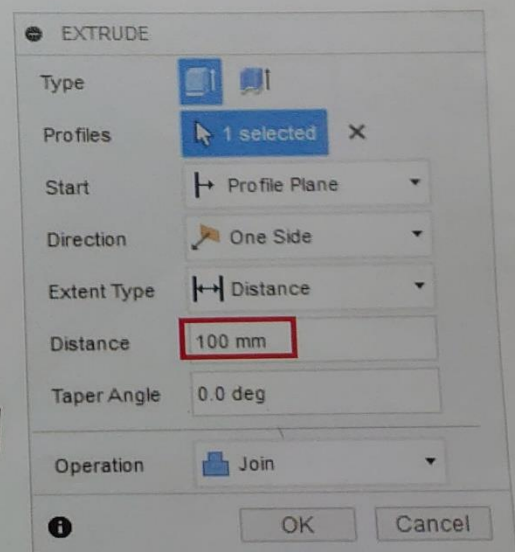
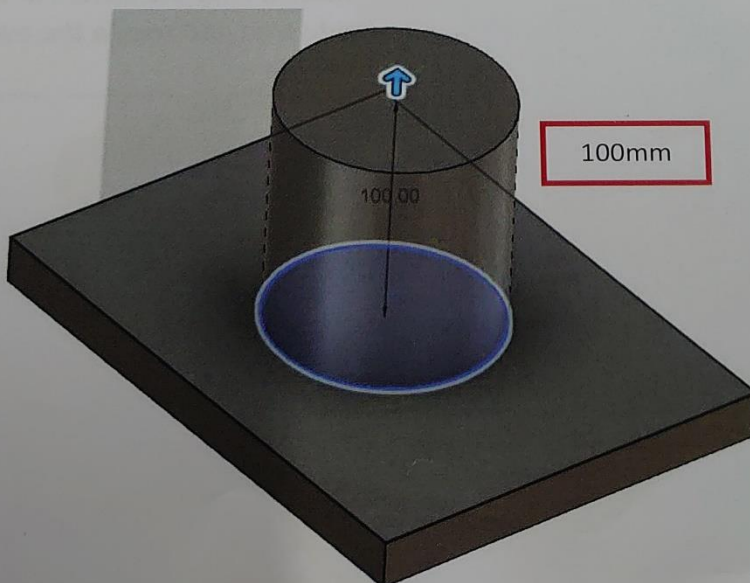


Your cylinder is now the correct diameter but not the correct height.  
Increase the height from 50mm to 100mm



To Modify the cylinder height from 50mm to 100mm

- 1- Right click the cylinder extrude icon
- 2- Select Edit Feature
- 3- Change dim value from 50mm to 100mm
- 4- Click OK





## 3D 360 CAD Training course worksheet exercises

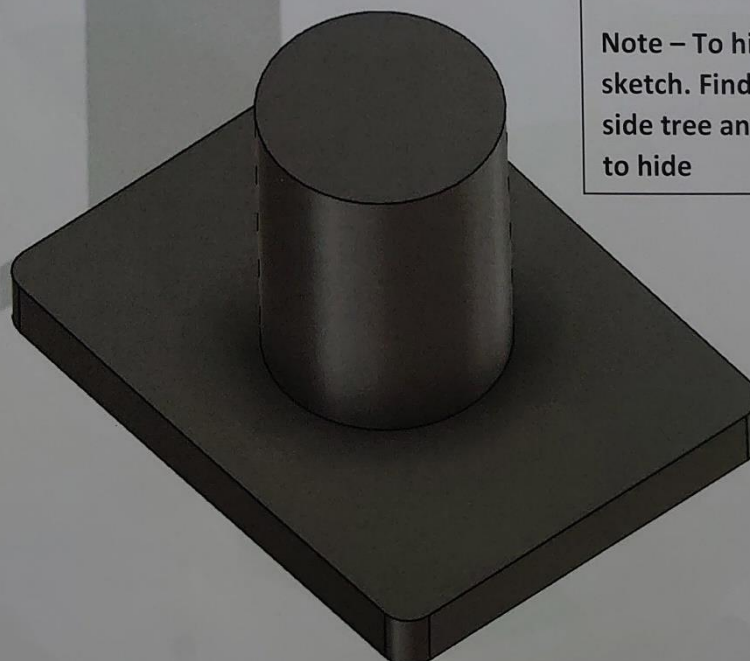
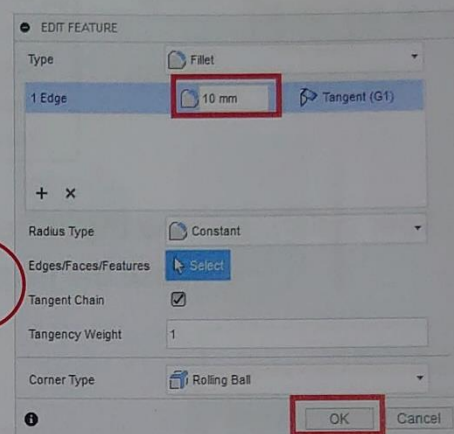
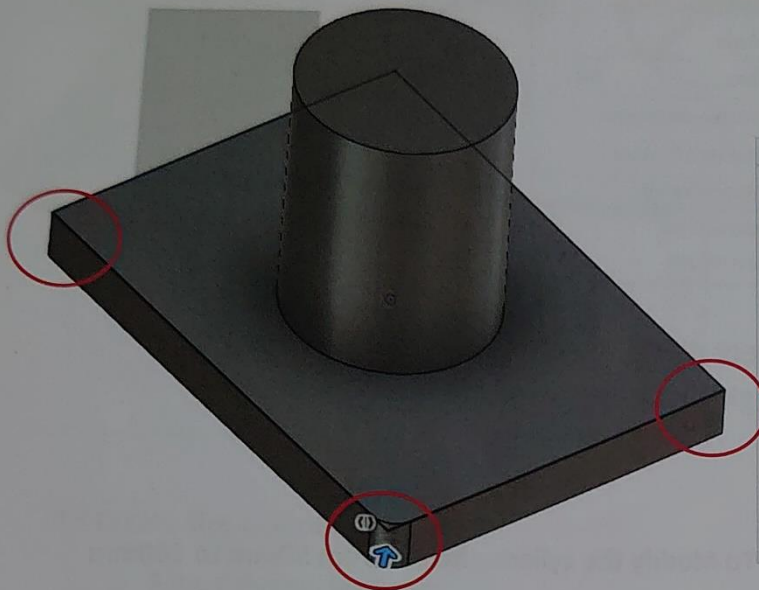


Step 5 – Add 10mm fillets to each for the 4 corners of the base solid.

Select the fillet tool



- Select one of the vertical corner lines.
- Next use the control key to select the remaining 3
- To set the size of your fillet type 10mm into the highlighted box

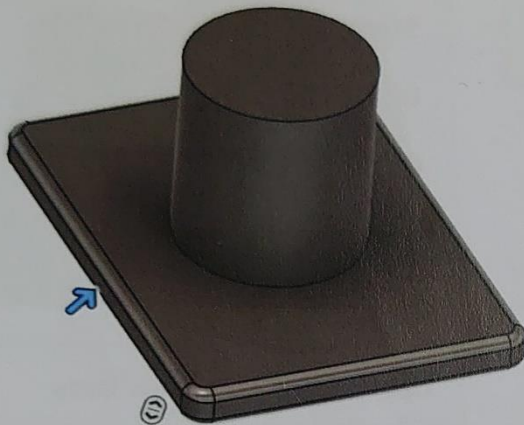


All 4 corners of the model now have 10mm rounded edges.

Note – To hide your original sketch. Find the sketch in the side tree and toggle the eyeball to hide

Step 6 – Using the Fillet command. Add 5mm rads to the top and bottom of the cube and a 15 mm rad to the joint between the cylinder and the square base.

Add 5 mm filets to the top and bottom



To Add the 15mm Rad

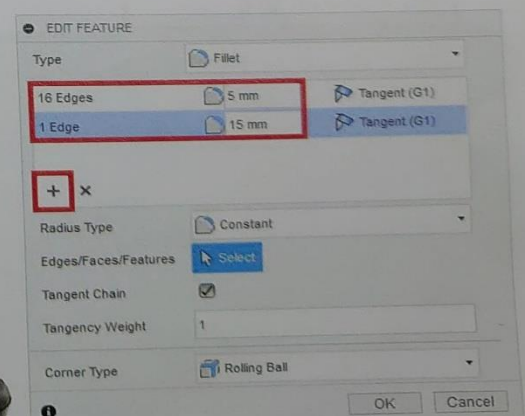
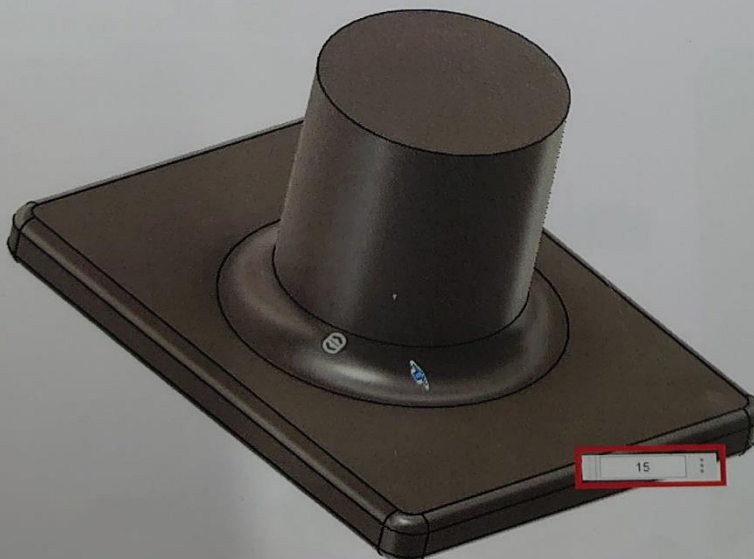
Click the + button



Add selection set.

Tip 1- To Add more differing sized fillets just keep using the +

Tip 2 – Don't try and add all the fillets in one go as this will add complexity to your model

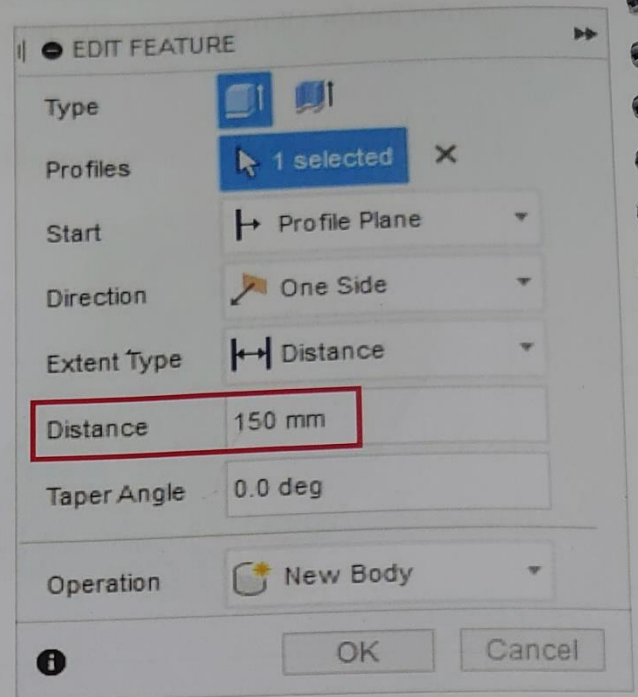
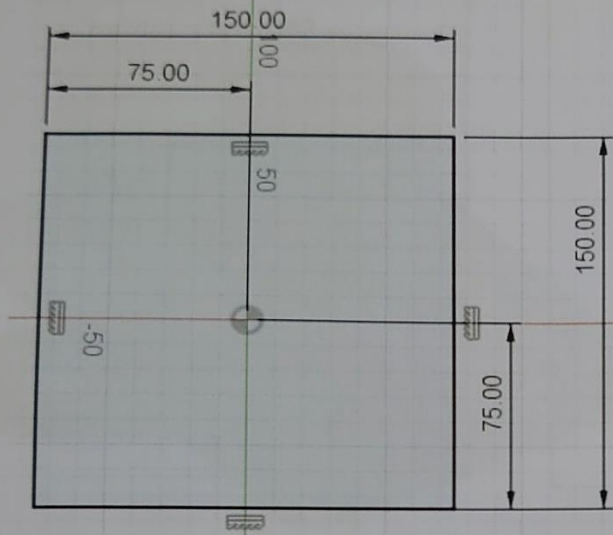


## 3D 360 CAD Training course worksheet exercises



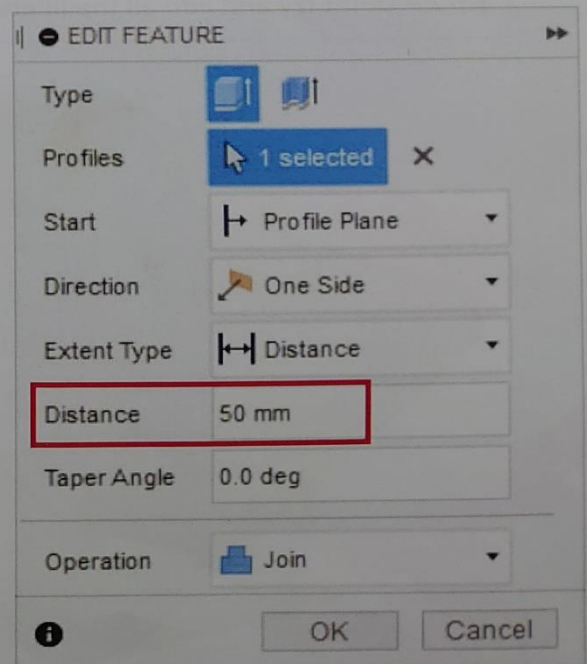
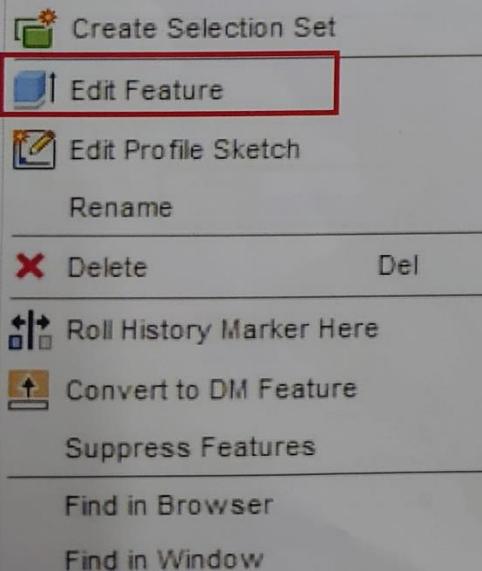
Step 7 – Modify cube back to a D= 150mm x W= 150mm x H= 150mm with an 80 Dia x 50 high cylinder on the top

Modify your original sketch back to a 150 x 150mm cube



Next Edit your cylinder extrude feature

Change the height to 50mm

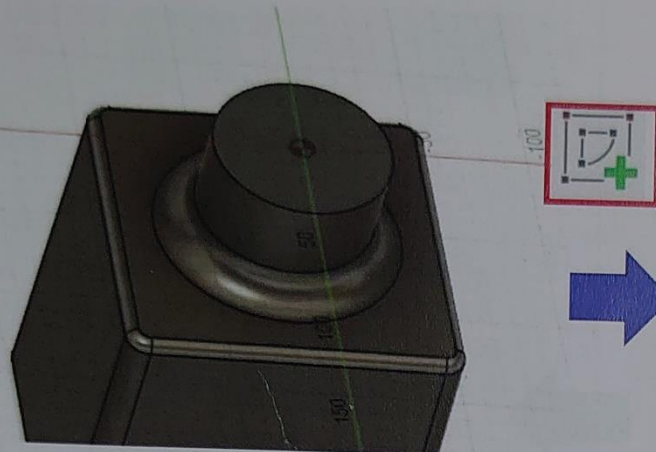




Step 8 – Cut a 50mm Dia hole 100mm into the center of the cylinder & Cube

Select the surface on top of the cylinder and click the Sketch Icon

Sketch a 50mm circle.



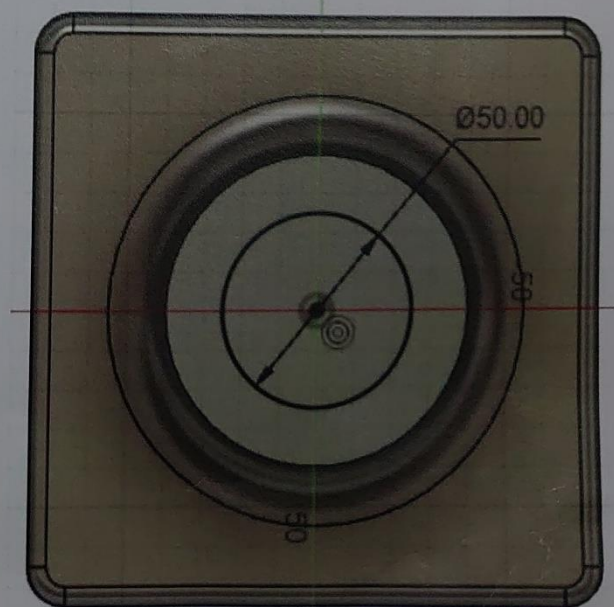
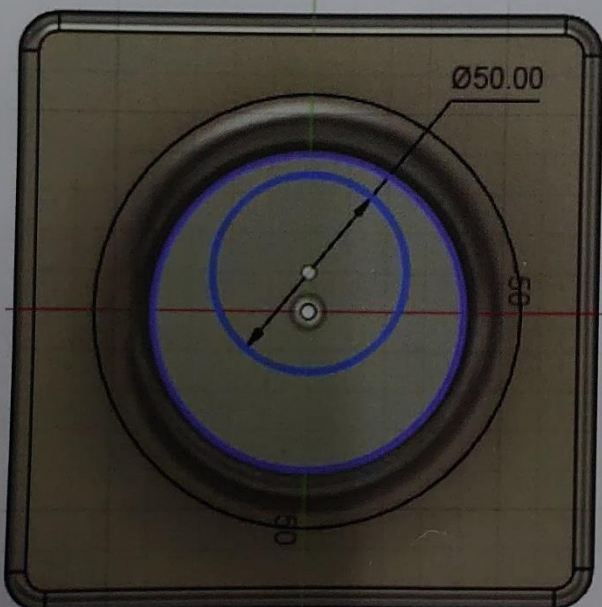
Use the concentric  
command



If the circle does not snap to the  
center.

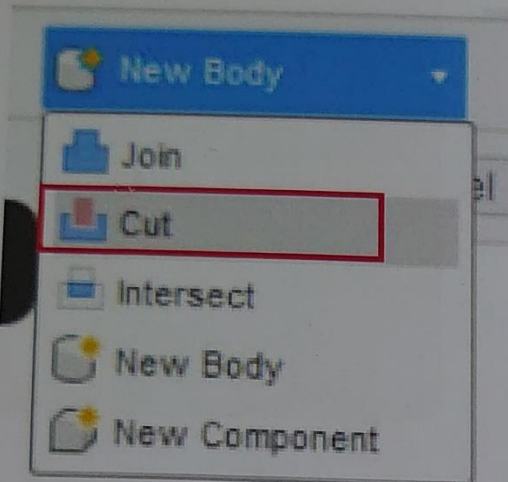
Use the Concentric constraint  
command to lock the circle position

Finished sketch should look like below.  
Note the Concentric constraint icon

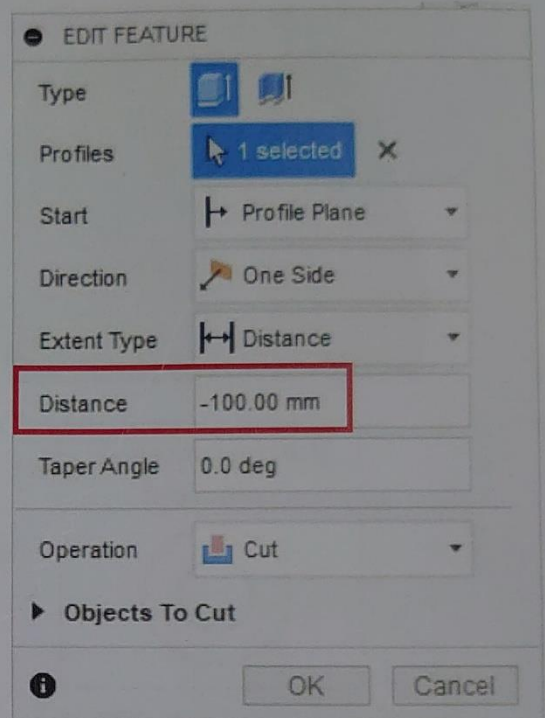


Step 8 – Cut a 50mm Dia hole 100mm into the center of the cylinder & Cube  
Continued

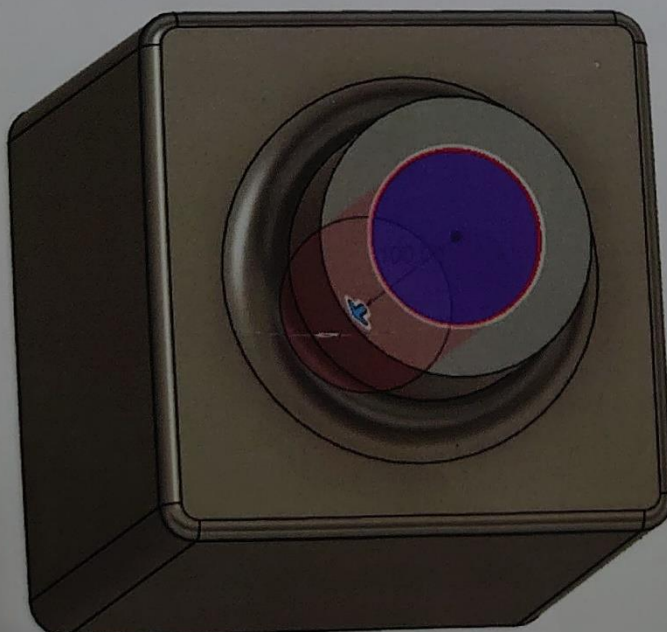
Select the cut option in the  
extrude tool



Set the distance to -100 mm



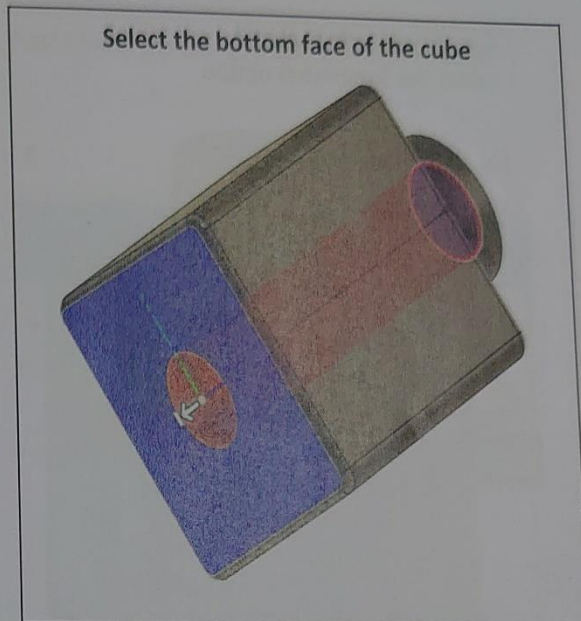
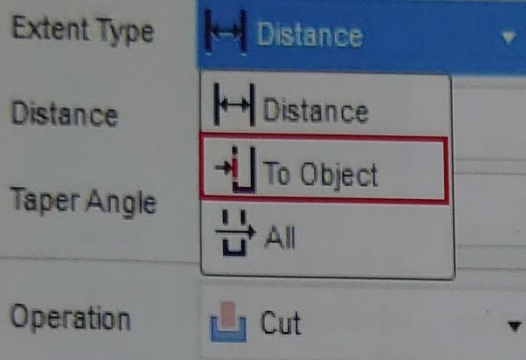
The Cut should look like the image below





Step 9 – Change the cut to run to the bottom face of the cube

Edit the extrude and set the extrude type 'to object'

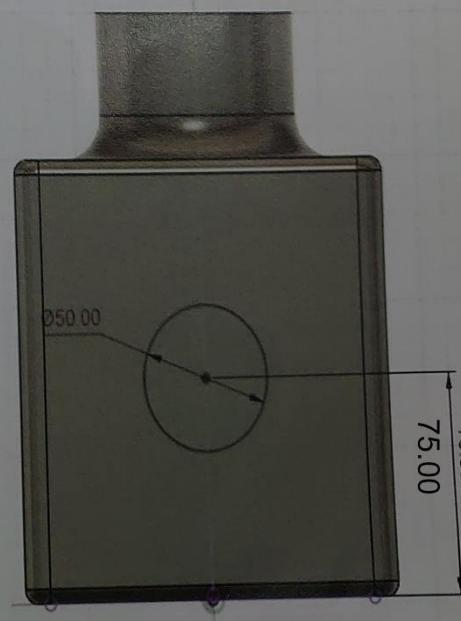
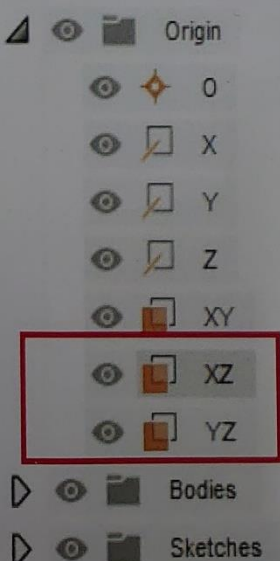


Step 10 – Creating a 50mm Dia cross extrusion on the center plane of the solid. Position the feature 75mm from the base of the cube

Select one of the middle planes to sketch on

Sketch a circle as shown.  
Snap the ctr to the ctr plane

Dimension the circle Dia to 50 mm and  
position 75mm up from the bottom face

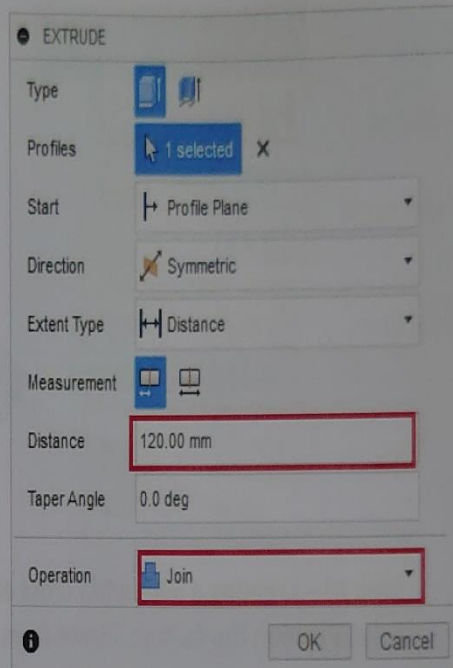
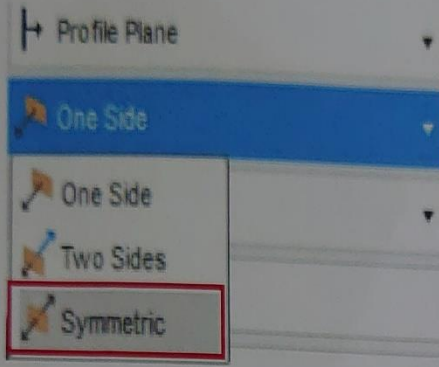




Step 10 – Creating a 50mm Dia cross extrusion on the center plane of the solid. Position the feature 75mm from the base of the cube – Continued

Use the extrude tool and  
select the Symmetric option

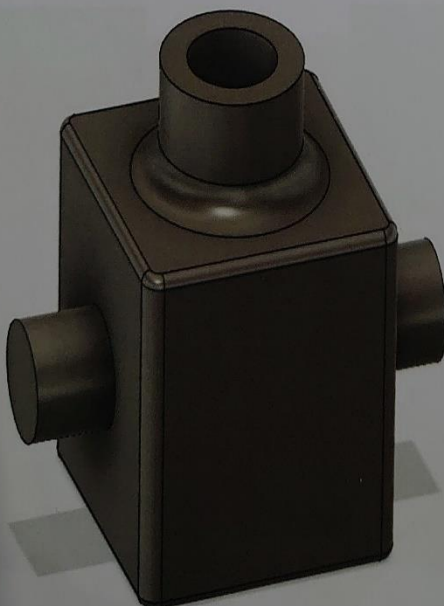
Set the distance to 120mm on each side



Remember to select  
the Join Option

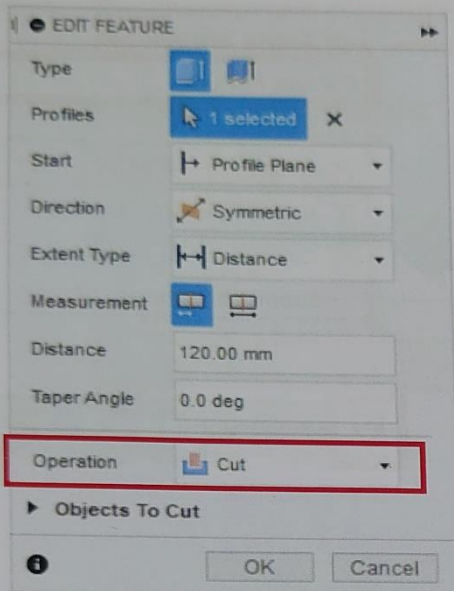


Your model should look like the image below



**Step 11 – Changing the 50mm diameter cross section 75mm from the base of the cube to a cut.**

Edit the extrude and make it a cut



Your Model should look like the image below



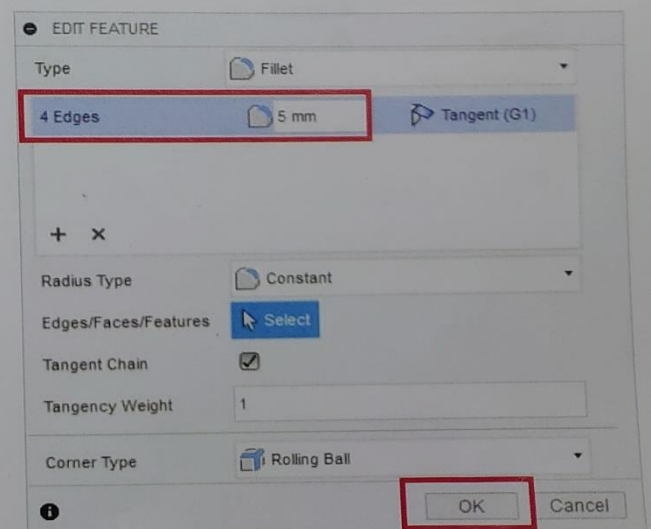
**Step 12 – Add 5 mm fillets to the edges of the 4 holes you have just created**

Your model should look like the image below

Select the Fillet command and pick the 4 edges to add the fillets



Set the size of the fillets to be 5mm



## 3D 360 CAD Training course worksheet exercises

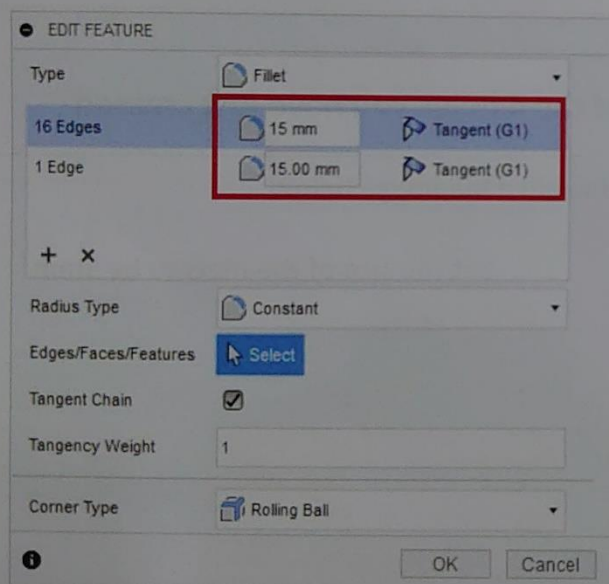
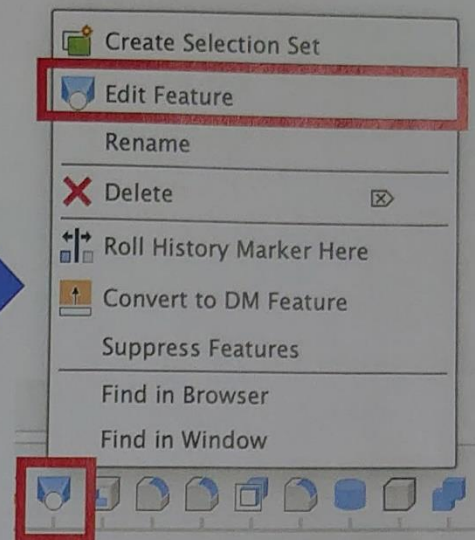
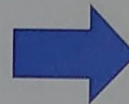


### Step 13 Continued – Increase your cubes 5mm fillets to 15mm

Select the Fillet feature from the model tree at the bottom left of the screen

Modify fillet sizes from 5mm to 15mm

- 5- Select the fillet feature from the model tree
- 6- Right click to open up the modification menu
- 7- Select Edit Feature
- 8- Modify Fillet values



All Fillets on the cube should be 15mm

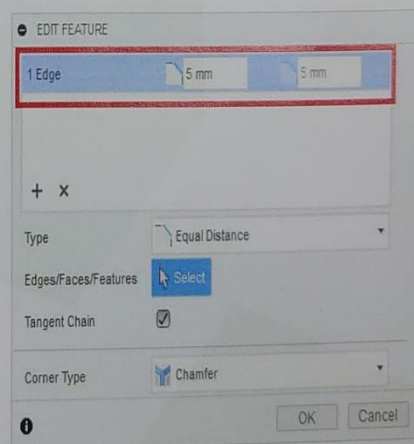
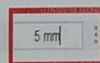
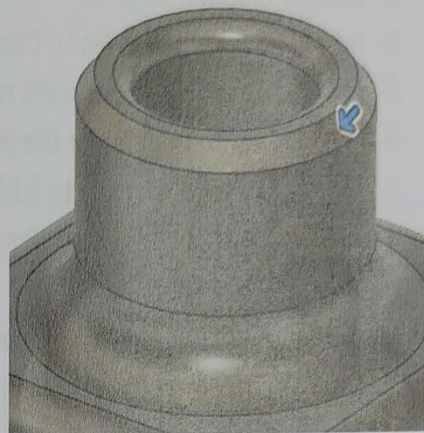
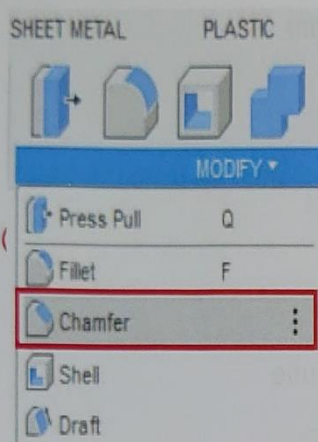




Step 13 – Adding a 5 x 5 mm chamfer or (45 degree) to the top edge of the cylinder.

Use the chamfer command  
The Chamfer icon is found on  
the modify pull down menu

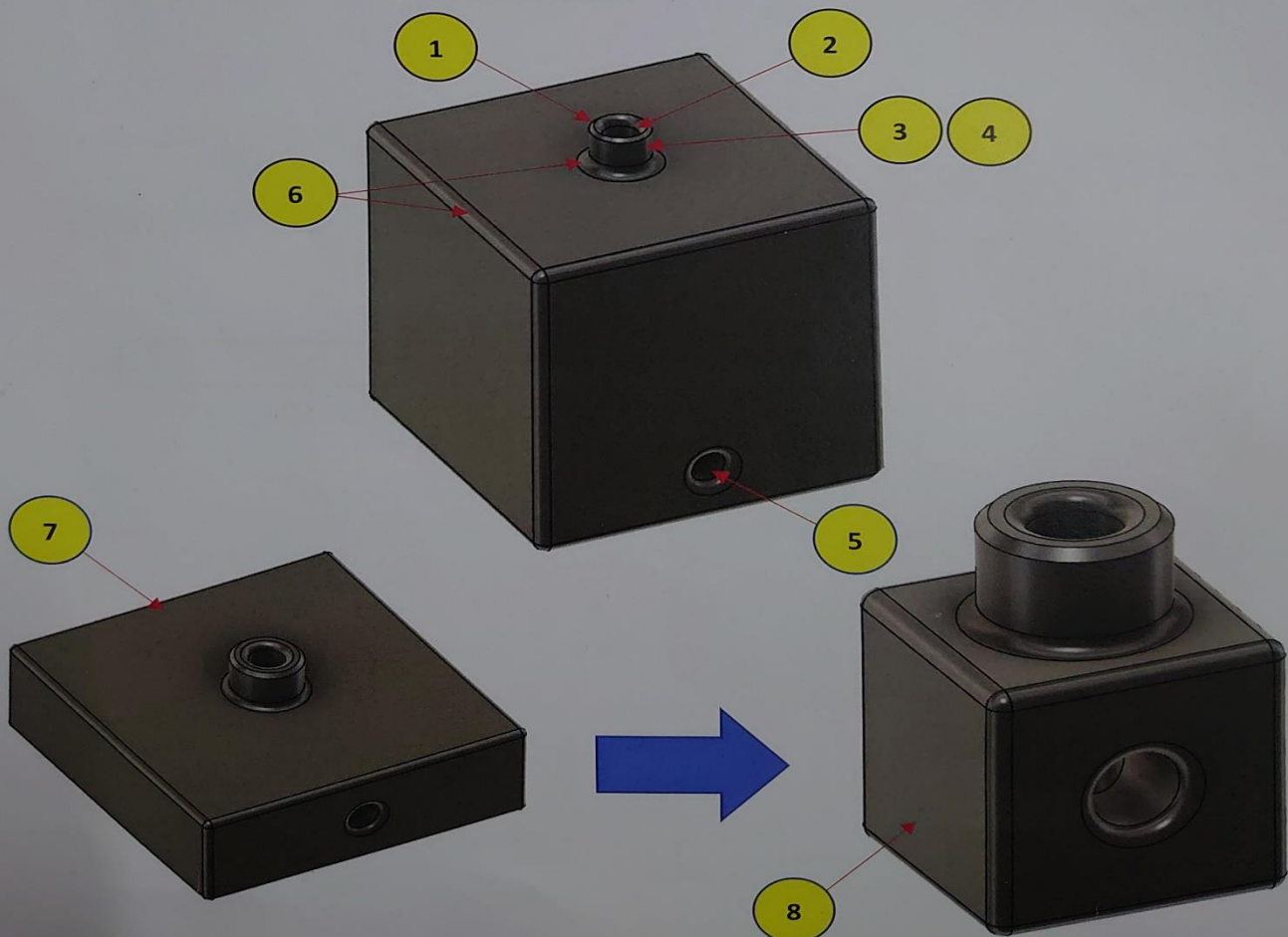
Select the outer edge of the cylinder



Step 14 – Modify the cube to a 3D printable size - 50x50x50mm with reduced hole diameters and reduced fillet and chamfer sizes

Modify in the following order

- 1- Reduce Chamfer to 2mm
- 2- Reduce the cylinder hole to Diameter to 15mm
- 3- Reduce the cylinder extrusion to 20mm
- 4- Reduce the outer cylinder Dia to 30mm
- 5- Reduce the through hole to a Dia of 15mm & 25mm up from the base
- 6- Reduce all Fillet sizes from 15mm to 3mm
- 7- Reduce Cube size Height to 50mm
- 8- Reduce Cube sketch to 50 x 50mm to finish cube



# FUSION 360 Advanced Training

3D 360 CAD Training course worksheet exercises



## Gear



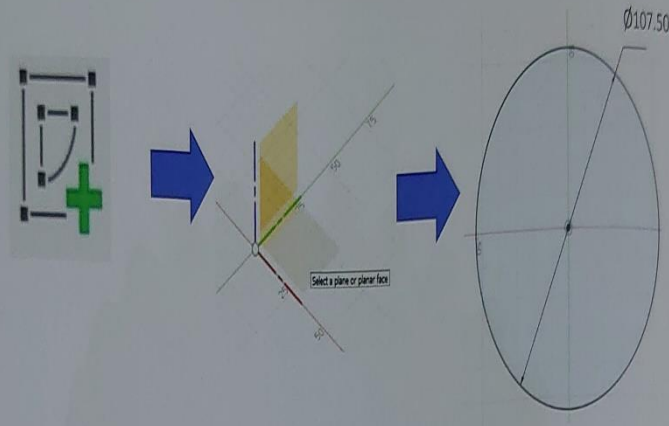


Step 1 – produce the body of the gear by extruding a 107.5mm Dia circle 40mm with a 30mm diameter hole through the center

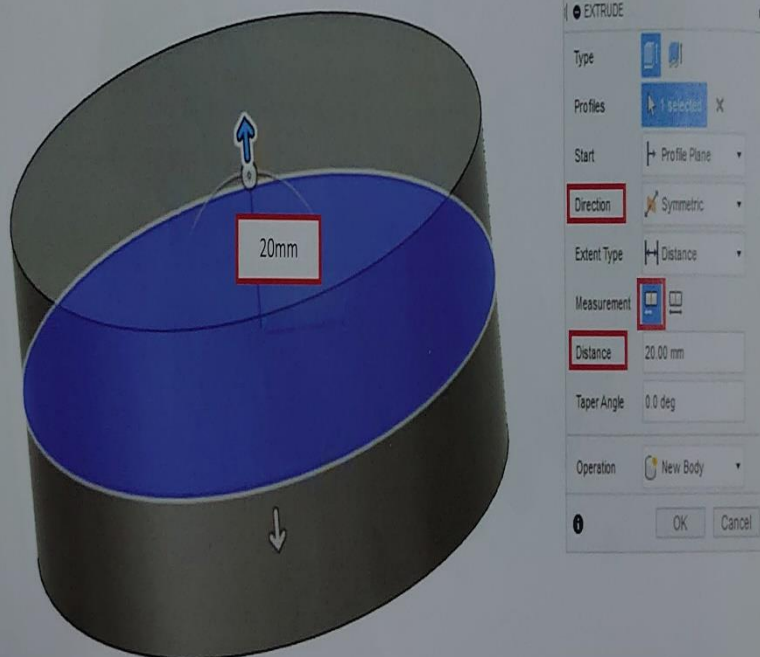
Select sketch icon

Select (top XY) plane

Sketch out a circle a 107.5mm Dia circle on the CTR point

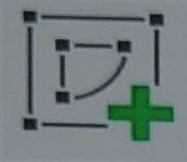


Extrude the sketch by 20mm on both sides using the symmetric direction function on the extrude tool

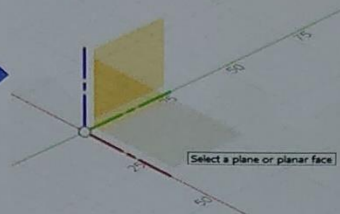


**Step 1 – produce the body of the gear by extruding a 107.5mm Dia circle 40mm with a 30mm diameter hole though the center**

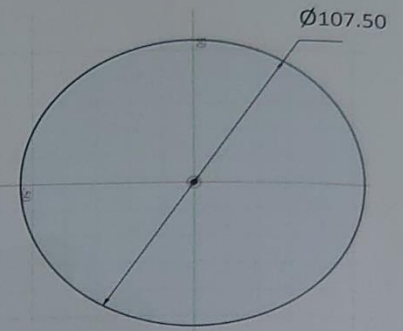
Select sketch icon



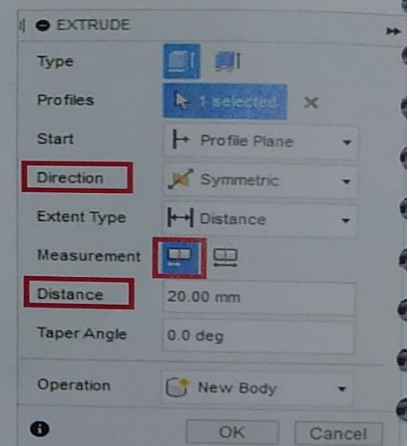
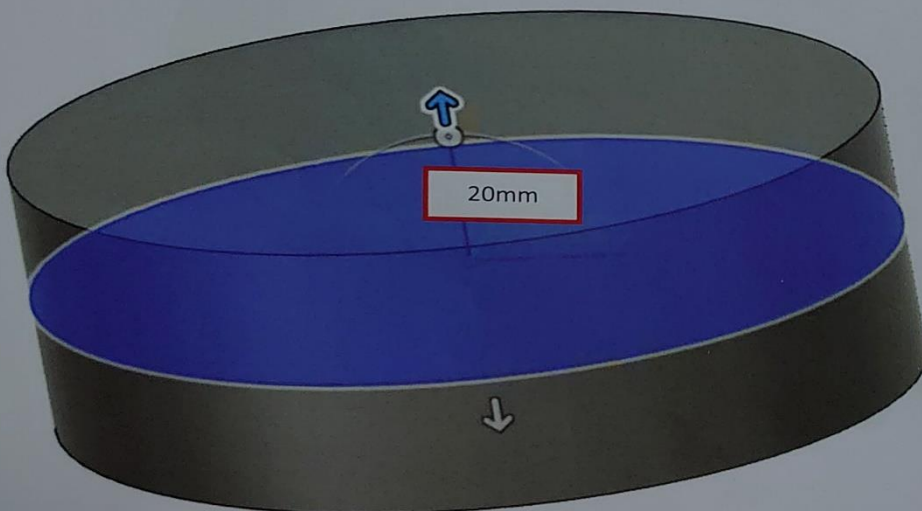
Select (top XY) plane



Sketch out a circle a 107.5mm Dia circle on the CTR point



**Extrude the sketch by 20mm on both sides using the symmetric direction function on the extrude tool**



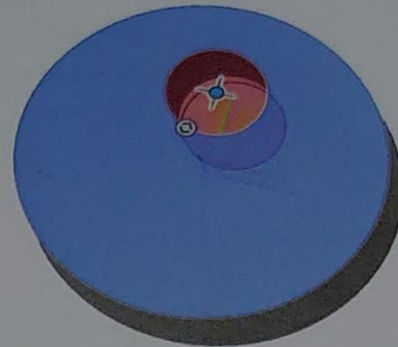
## 3D 360 CAD Training course worksheet exercises



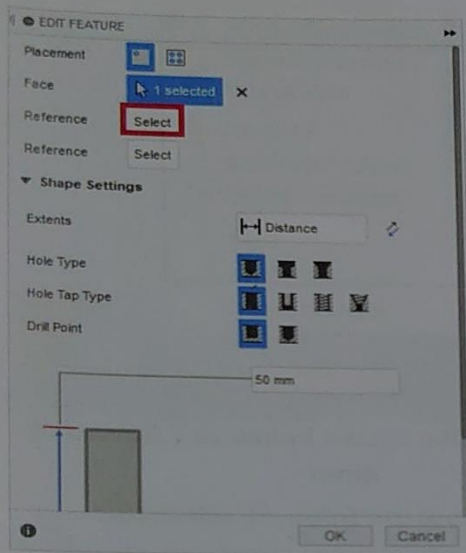
Select hole icon



Select the top face



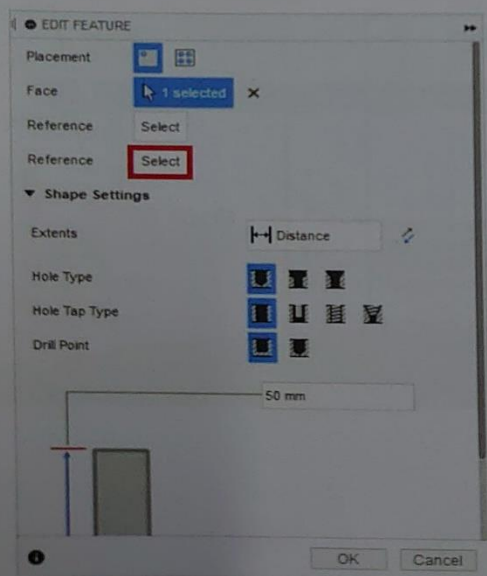
Select the first reference



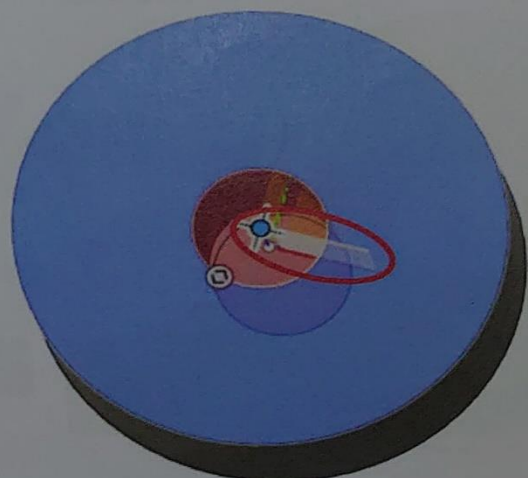
Select YZ plane



Select the second reference

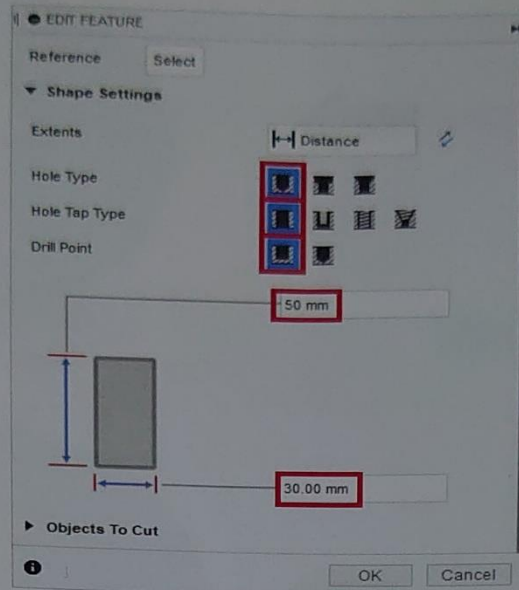


Select XZ plane





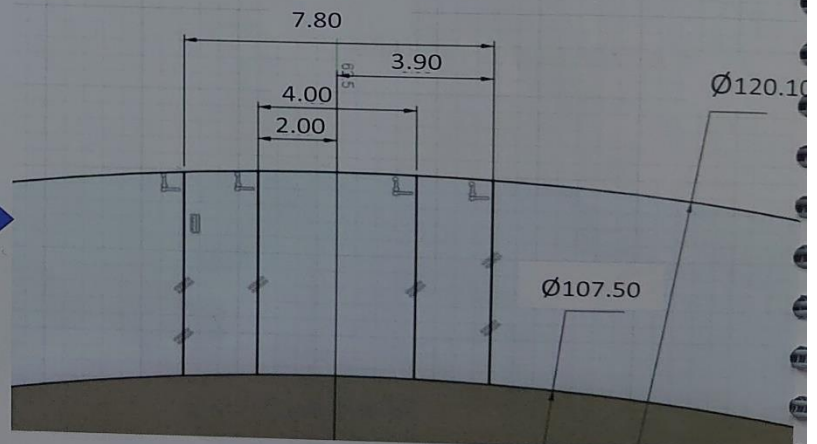
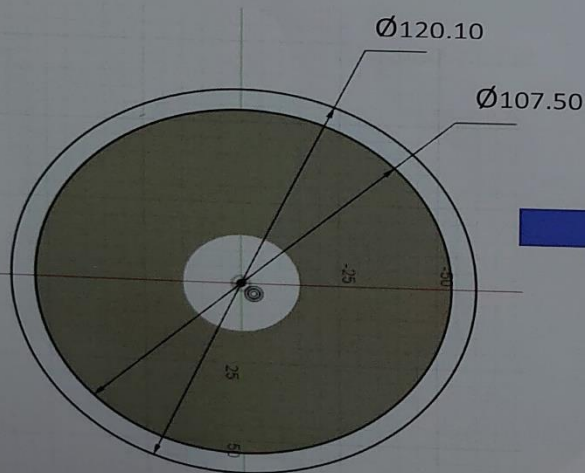
Set the depth to 50mm and the diameter to 30mm. Then select the simple hole type and simple hole tap type with a flat drill point



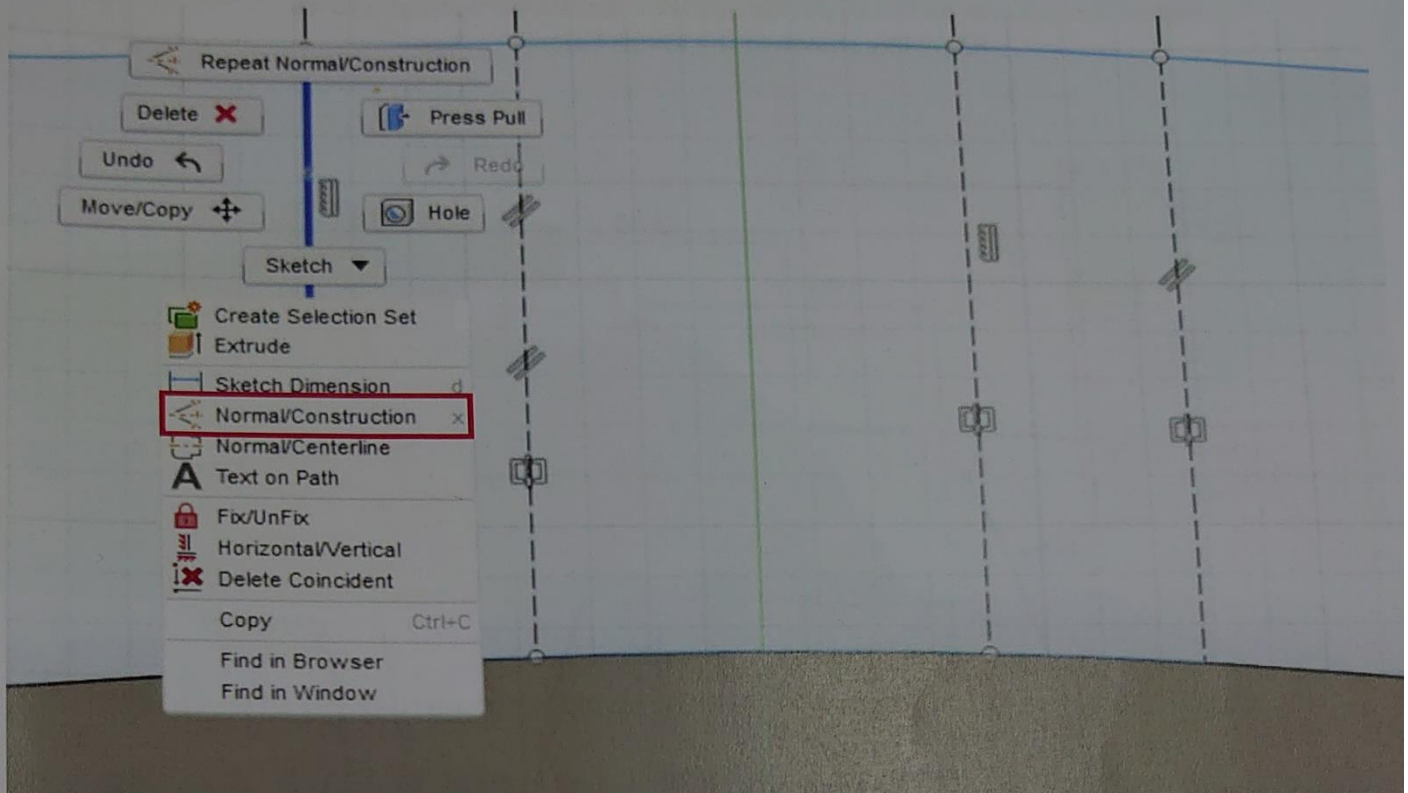
## Step 2 – produce the first tooth of the gear

On the top XY plane sketch out 2 circles on the CTR point a 107.5mm Dia circle and a 120.1mm Dia circle

Sketch out 4 lines in between the circles and dimensioned like shown below at 7.8mm and 4mm

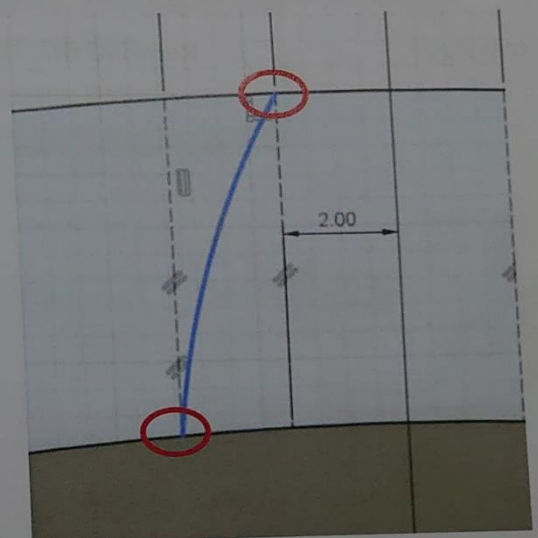
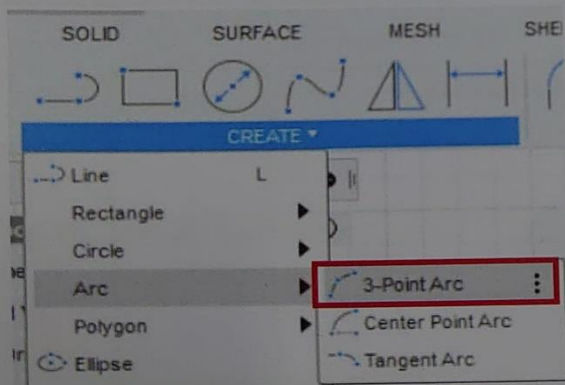


Set the 4 lines to be construction lines like shown below. Right click the line and select the Normal / Construction option

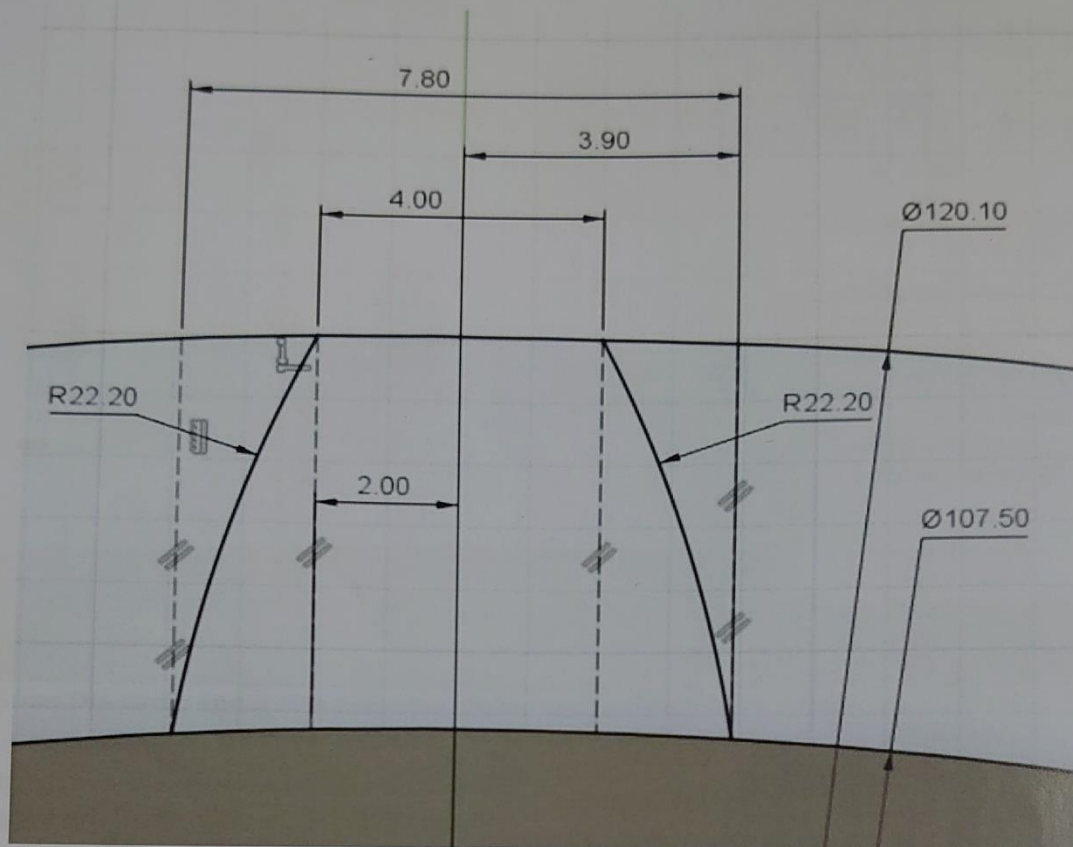


Select the 3 – point arc tool

Sketch the ark in-between the construction lines and the circle the like shown in the image below.



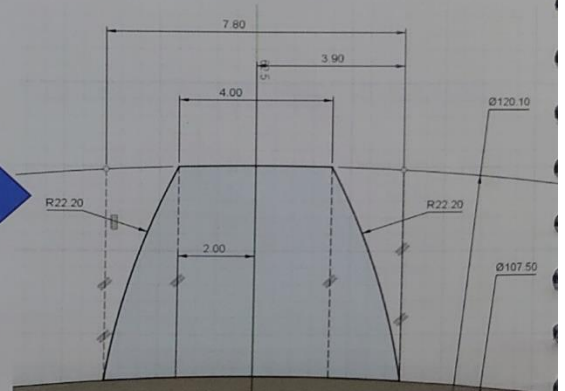
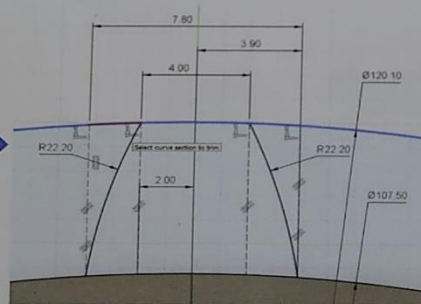
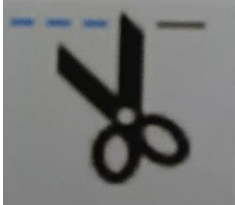
Repeat this process on the opposite side then dimension both arcs to be R22.2 like shown in the image below.



Select trim tool

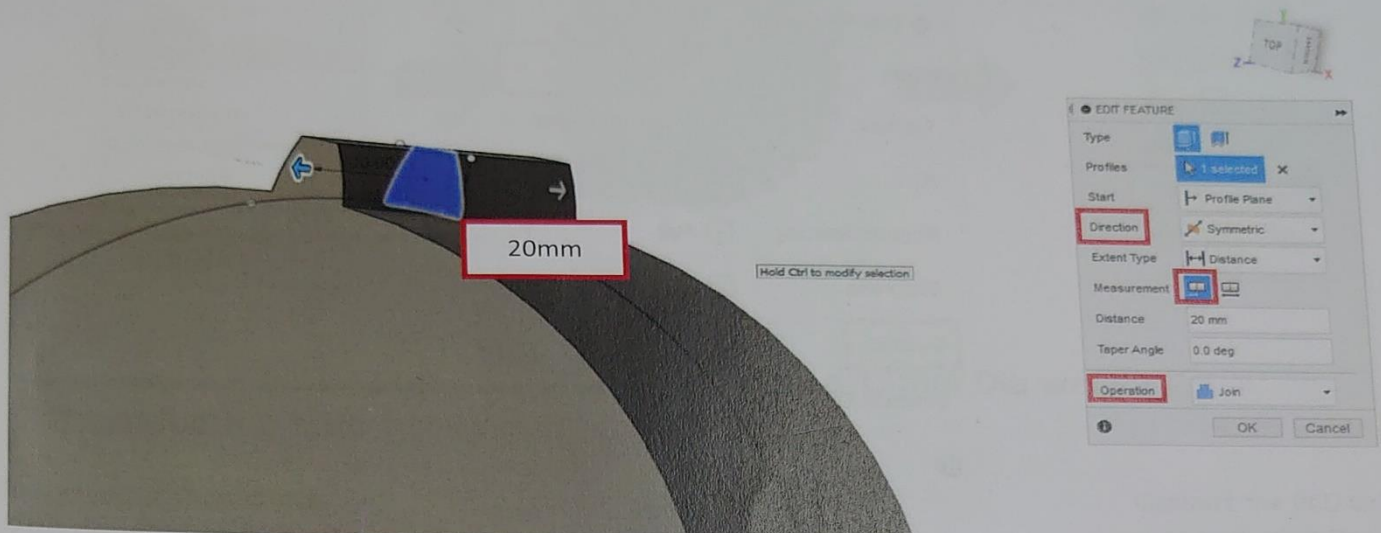
Remove the excess lines

it should look like the image below



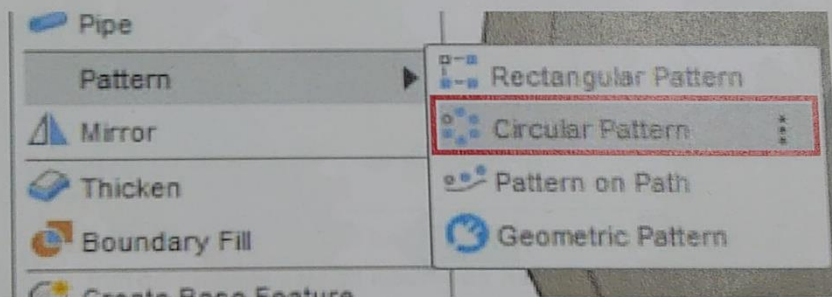


extrude the sketch by 20mm on both sides using the symmetric option with the join command



Step 3 – Use the pattern tool to add 30 teeth equally spaced around the gear.

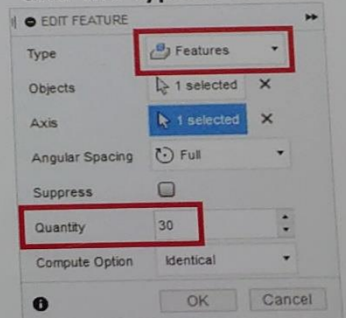
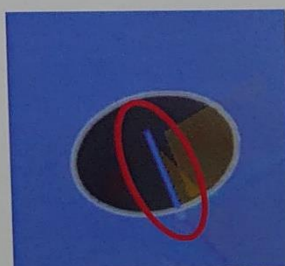
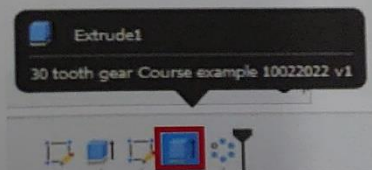
In the circular pattern tool select the previous extrude



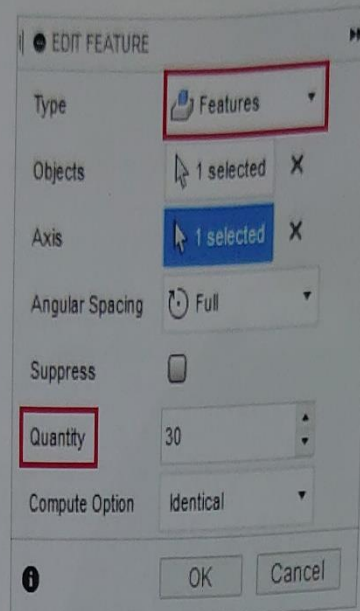
In the circular pattern tool select the previous extrude

Select the CTR line as the axis

Set the quantity to be 30 and the type to features



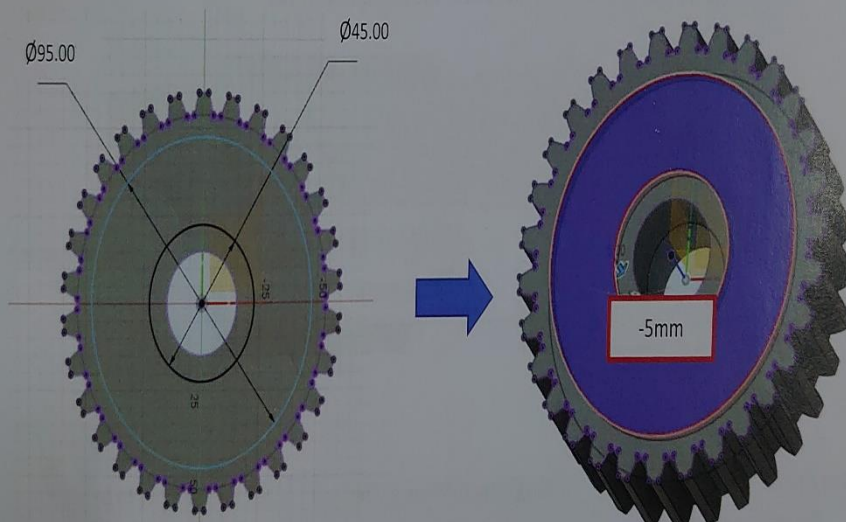
The feature box should look like  
the box below



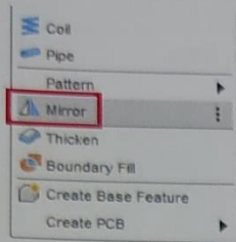
#### Step 4 – Reducing the side profile of the gear

On the top face of the gear sketch out 2  
circles a 45mm Dia and a 95mm Dia circle

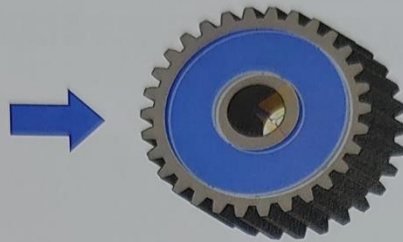
Extrude using the cut function - 5mm



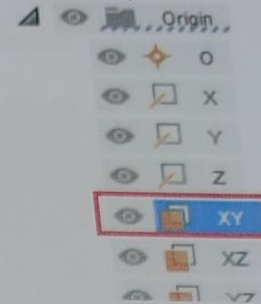
Use the mirror tool



Select the previous extrude

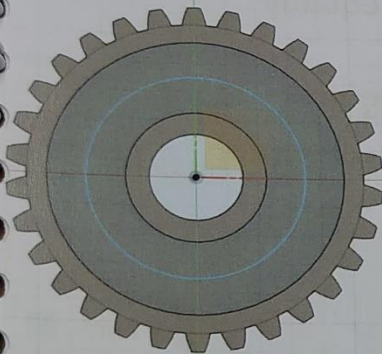


Select the CTR - XY plane as the mirror plane

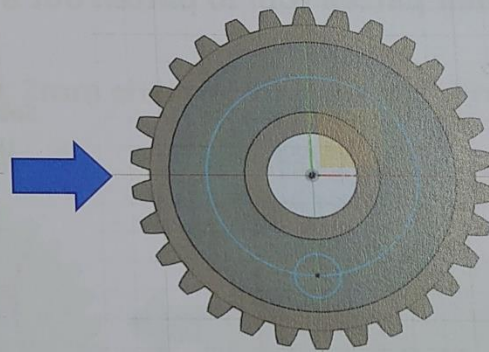


## Step 5 – Setting up the 70mm dia PCD and 15mm Dia weight reduction hole

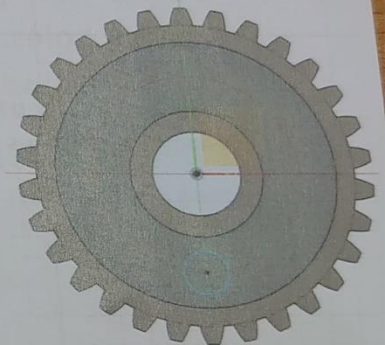
On the extruded face sketch out a PCD using the circle tool



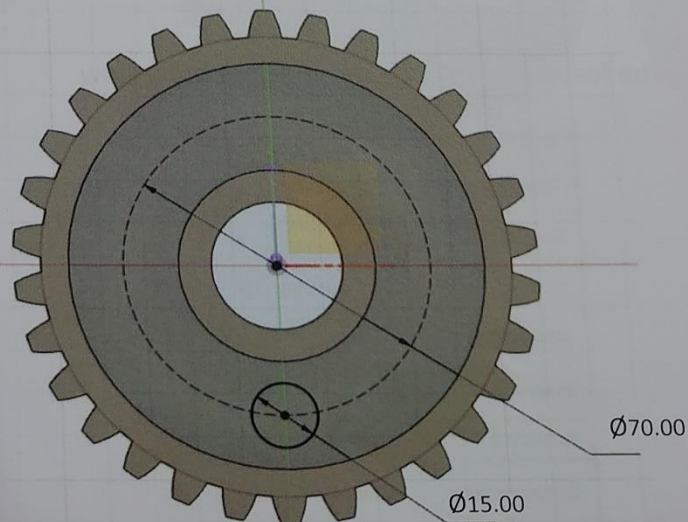
Sketch a smaller circle on the PCD and the CTR line



Convert the PCD to a construction line

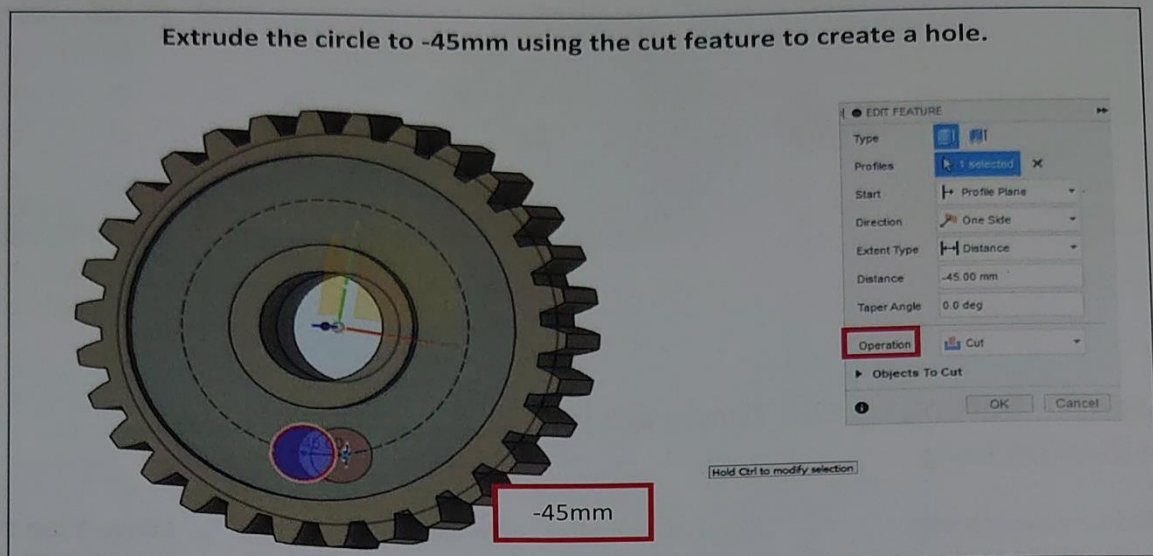


Dimension the sketch with the PCD being 70mm Dia and the circle being 15mm Dia. The sketch should now look like the image below



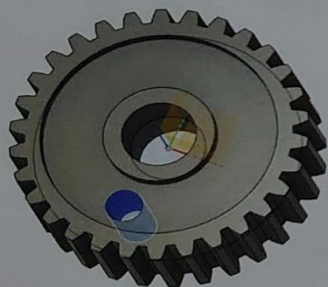


Extrude the circle to -45mm using the cut feature to create a hole.

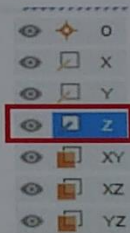


**Step 6 – Using the circular pattern tool to pattern out 8 equally spaced holes.**

Using the circular pattern tool select the previous extrude



Select the Z axis for the axis



Set the quantity of the pattern to 8

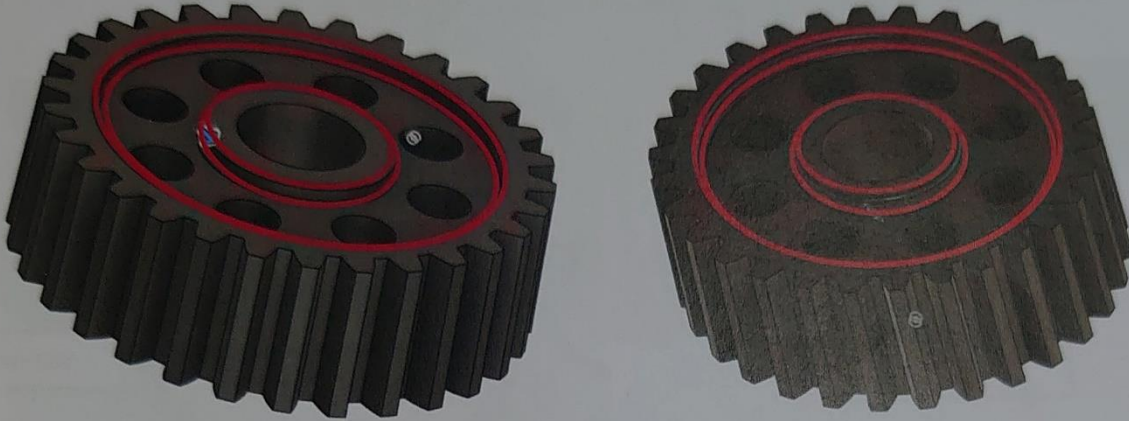


The feature box should look like the box below



### Step 7 – Adding 1mm fillets to the inner edges of the gear

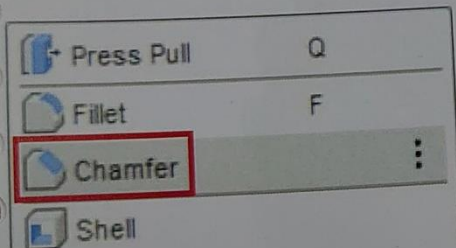
Use the fillet tool to add 1mm fillets each inner edge of the gear like shown below



### Step 8 – Adding 2mm chamfer to the weight reduction holes

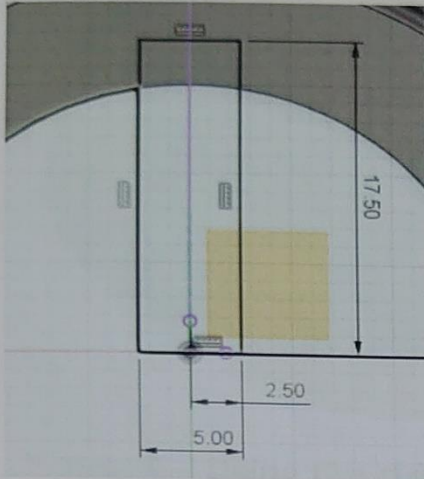
Select the chamfer tool

Add a 2 X 2mm equally distanced chamfer to the top and bottom of each hole on both sides of the gear

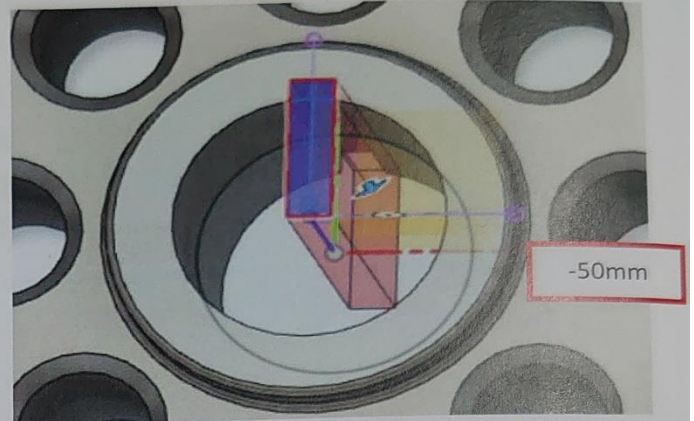


### Step 9 – Creating a 5mm keyway in the gear

Sketch out a 17.5mm by 5mm rectangle on the top face of the gear

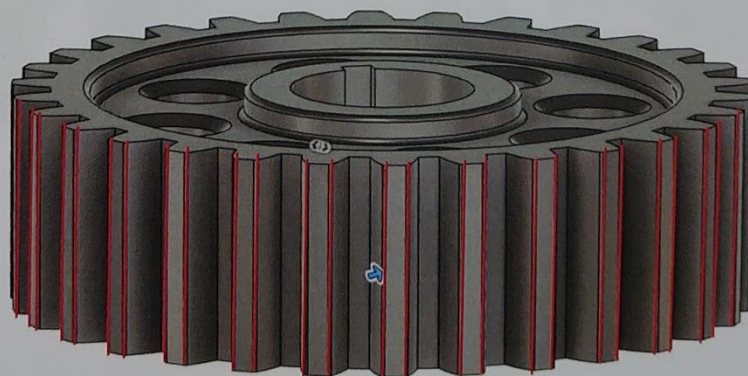


Extrude the rectangle down -50mm using the cut feature



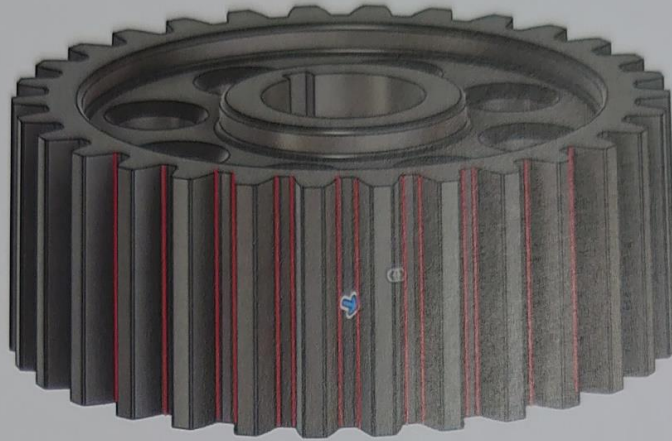
### Step 10 – Adding the final fillets to the teeth of the gear

Use fillet tool add a 0.5mm fillet to each outer edge of the gear teeth





Use fillet tool add a 1 mm fillets to each inner edges of the gear teeth



The gear should now look like the image below

