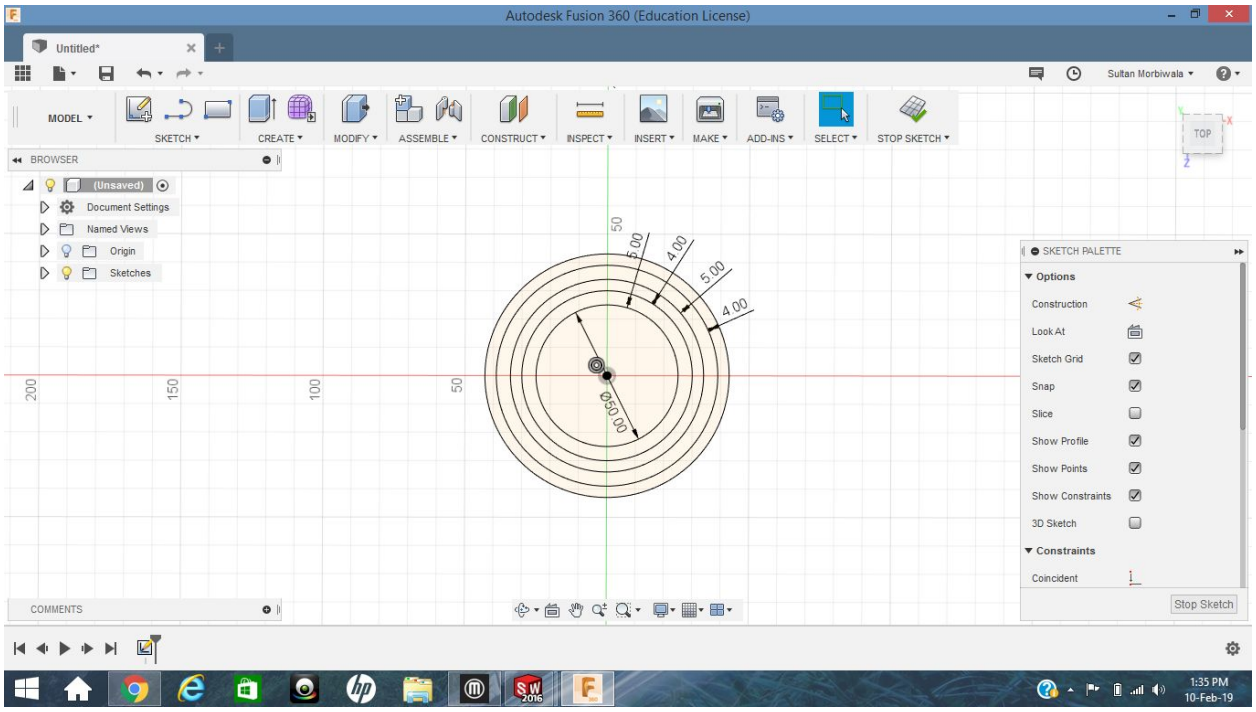
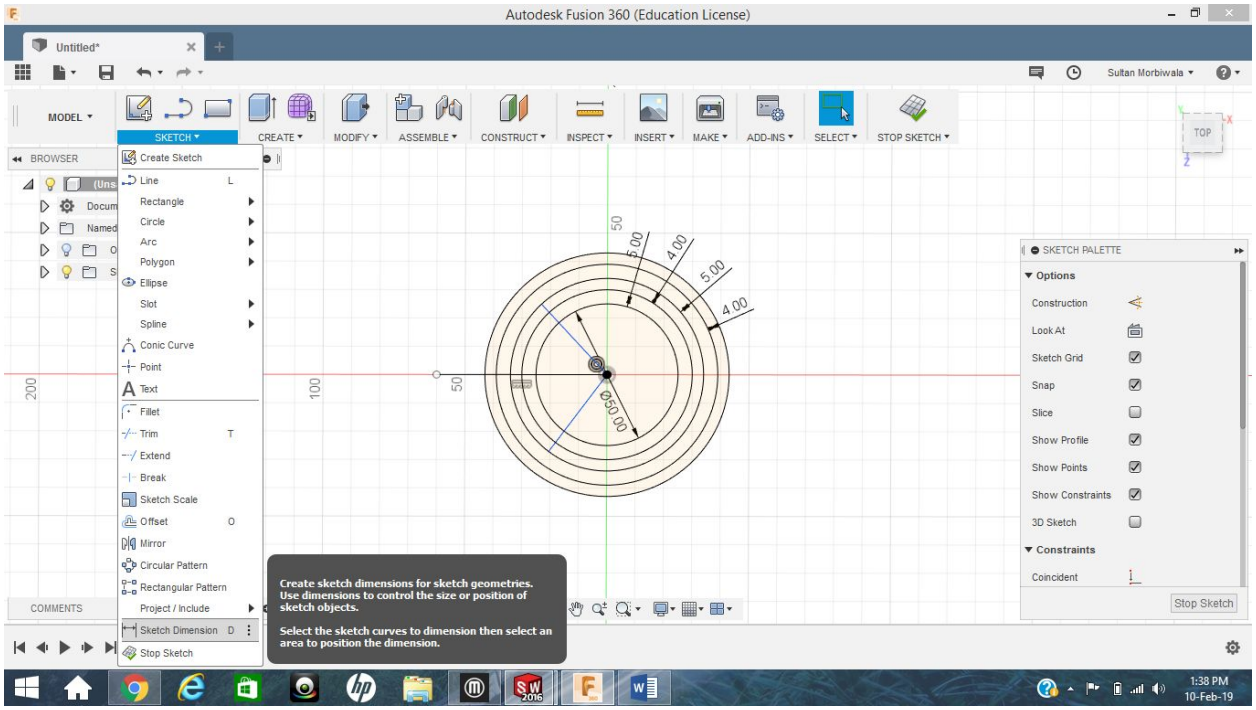
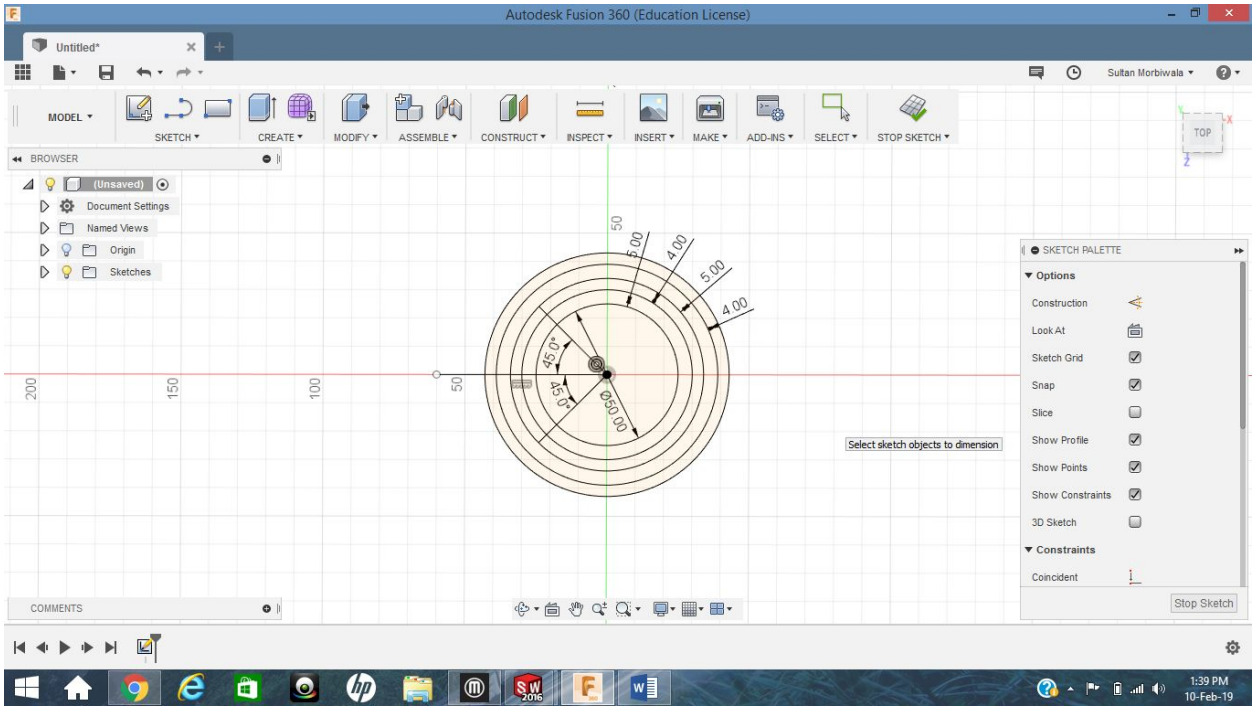
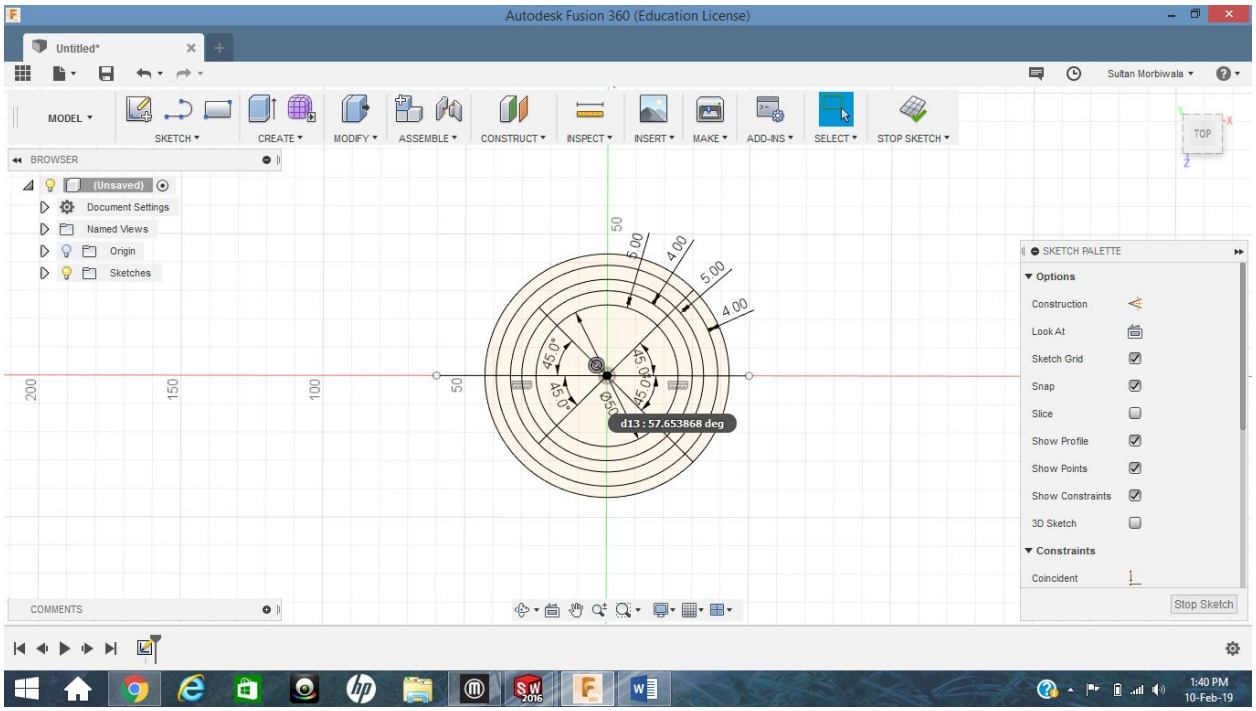
Step 1:- Make a circle of 50mm diameter in beginning and create offset of 4 & 5mm like the image below



Step 2:- Draw lines of 45 degree from the horizontal line on both sides




Autodesk Fusion 360 (Education License)
Untitled*
Sultan Morbiwala
MODEL
SKETCH
CREATE
MODIFY
ASSEMBLE
CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
STOP SKETCH
BROWSER
(Unsaved)
Document Settings
Named Views
Origin
Sketches
TOP
SKETCH PALETTE
Options
Construction
Look At
Sketch Grid
Snap
Slice
Show Profile
Show Points
Show Constraints
3D Sketch
Constraints
Coincident
Stop Sketch
5.00
4.00
5.00
4.00
45.0°
45.0°
Ø50.00
Select sketch objects to dimension
COMMENTS
1:39 PM
10-Feb-19


Autodesk Fusion 360 (Education License)
Untitled*
Sultan Morbiwala
MODEL
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CREATE
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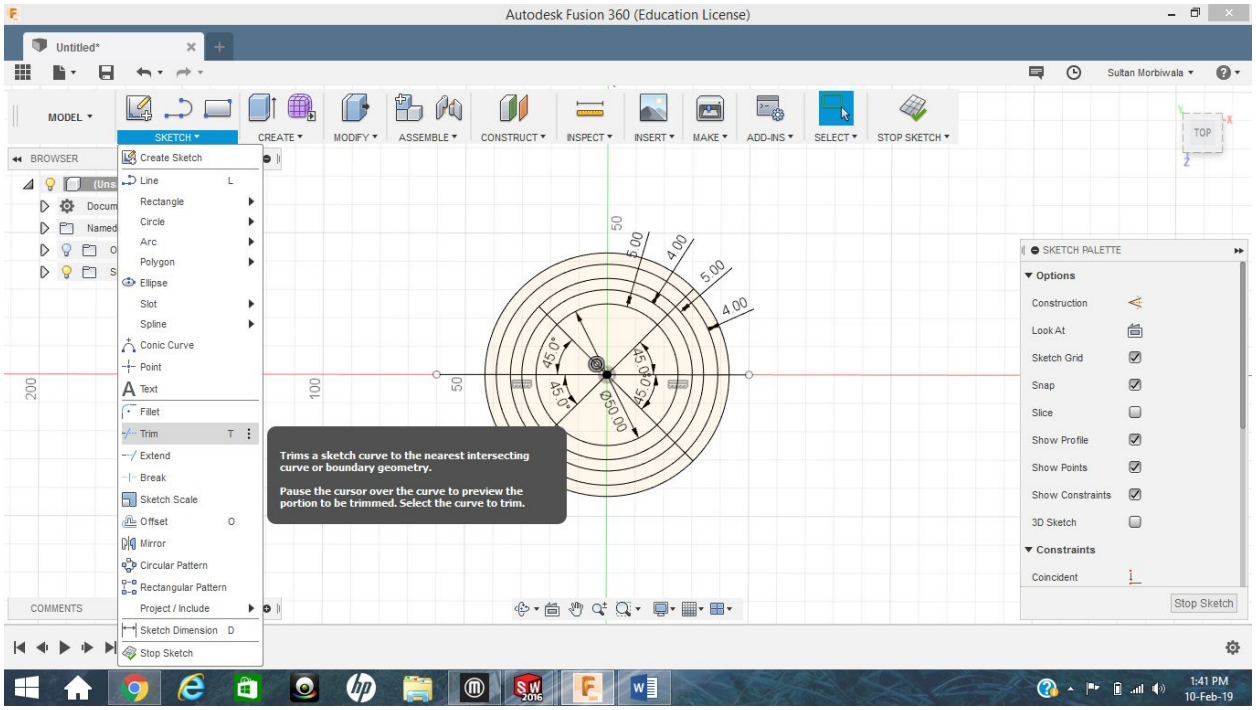
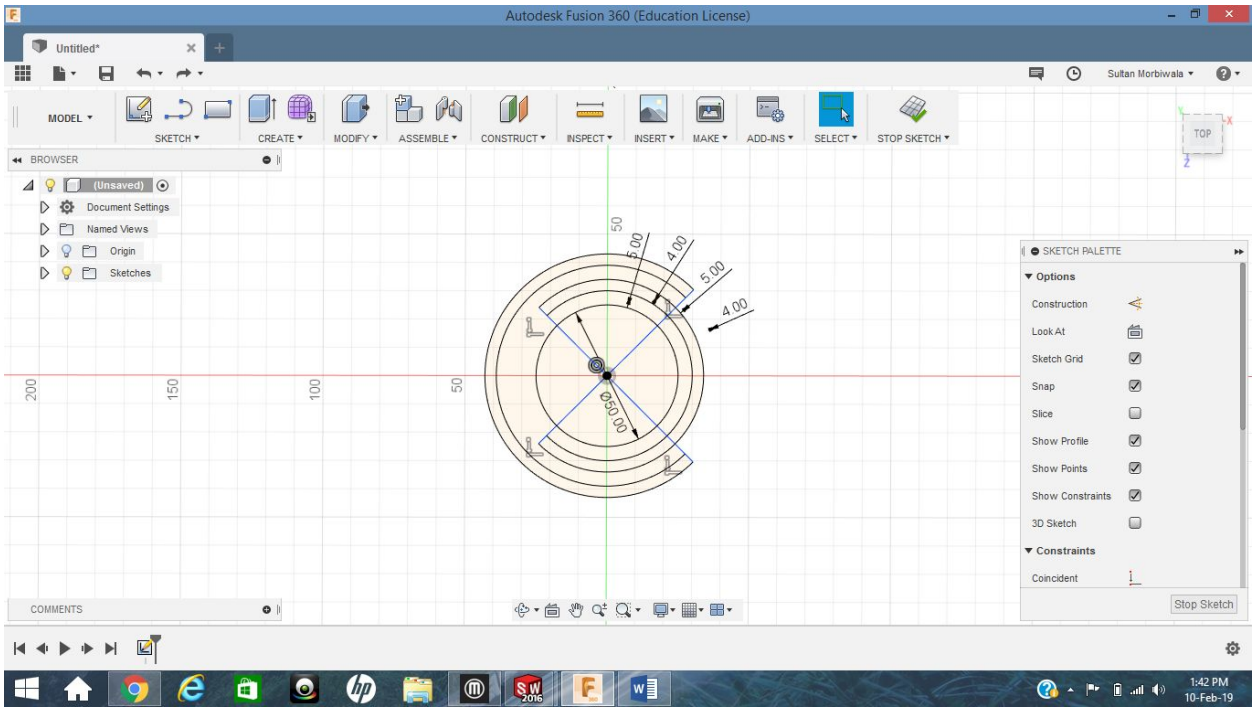
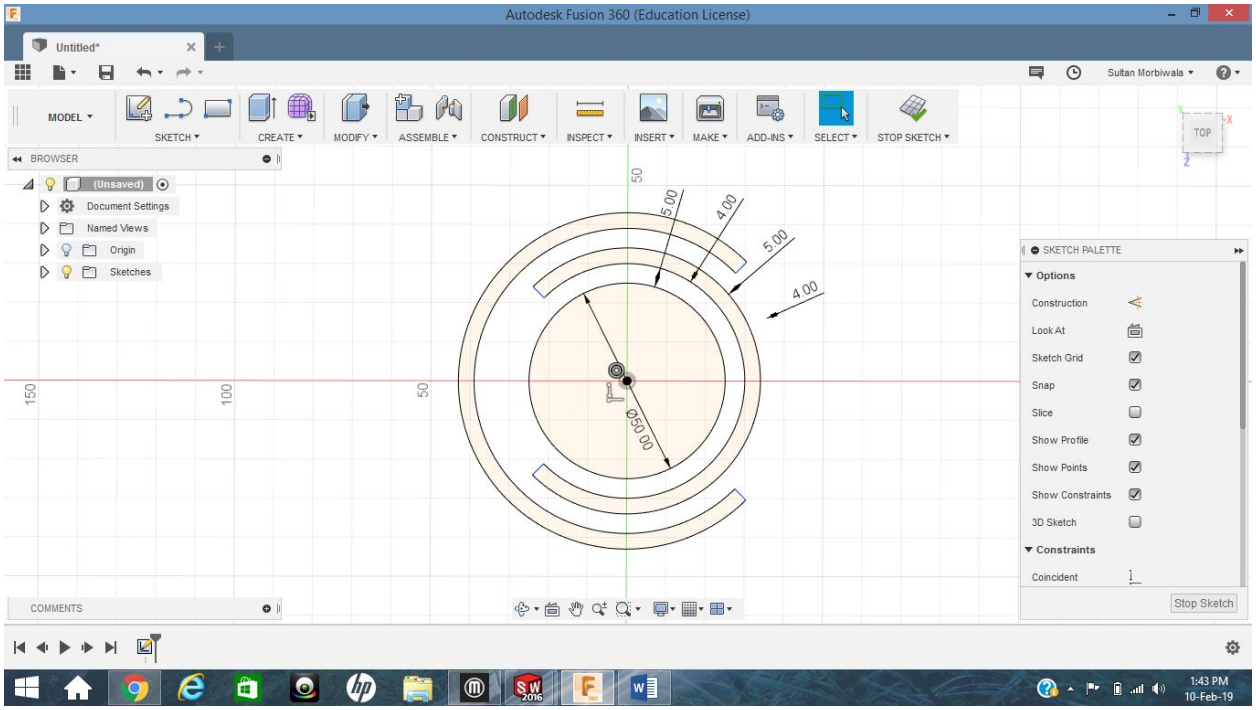
INSERT
MAKE
ADD-INS
SELECT
STOP SKETCH
BROWSER
(Unsaved)
Document Settings
Named Views
Origin
Sketches
TOP
SKETCH PALETTE
Options
Construction
Look At
Sketch Grid
Snap
Slice
Show Profile
Show Points
Show Constraints
3D Sketch
Constraints
Coincident
Stop Sketch
5.00
4.00
5.00
4.00
45.0°
45.0°
45.0°
d13 : 57.653868 deg
COMMENTS
1:40 PM
10-Feb-19

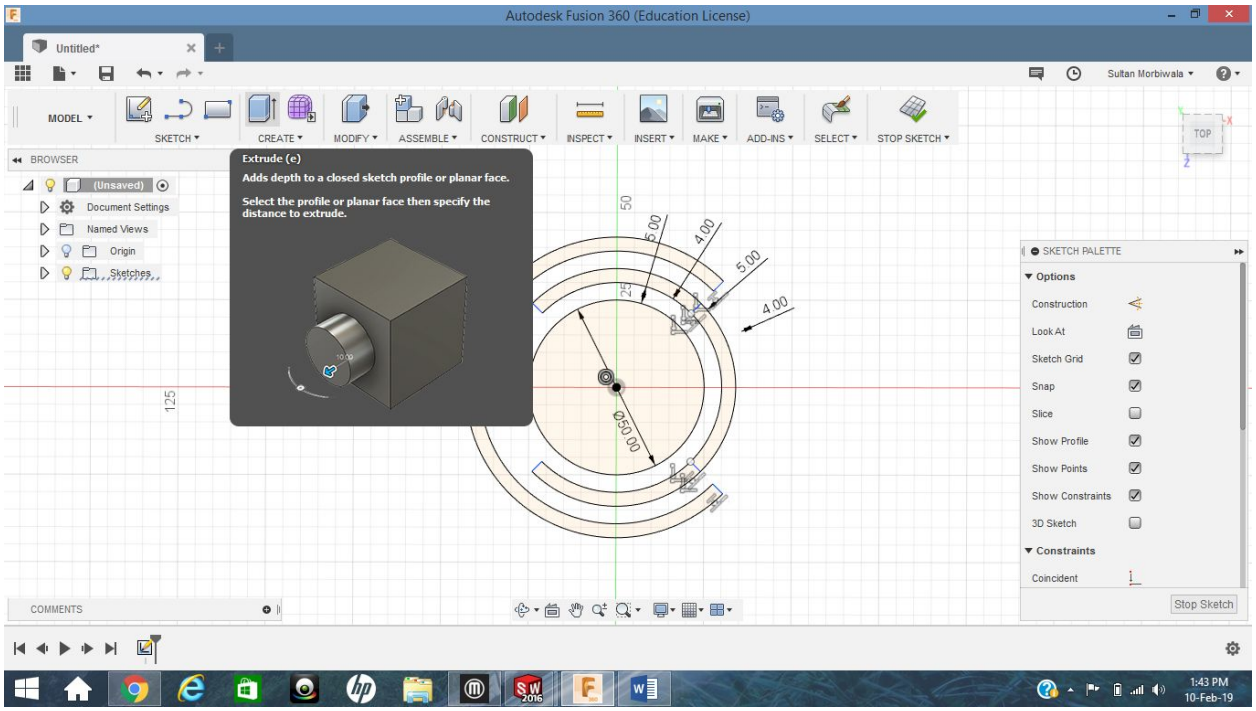
Step 3:- Use trim function to remove excess lines

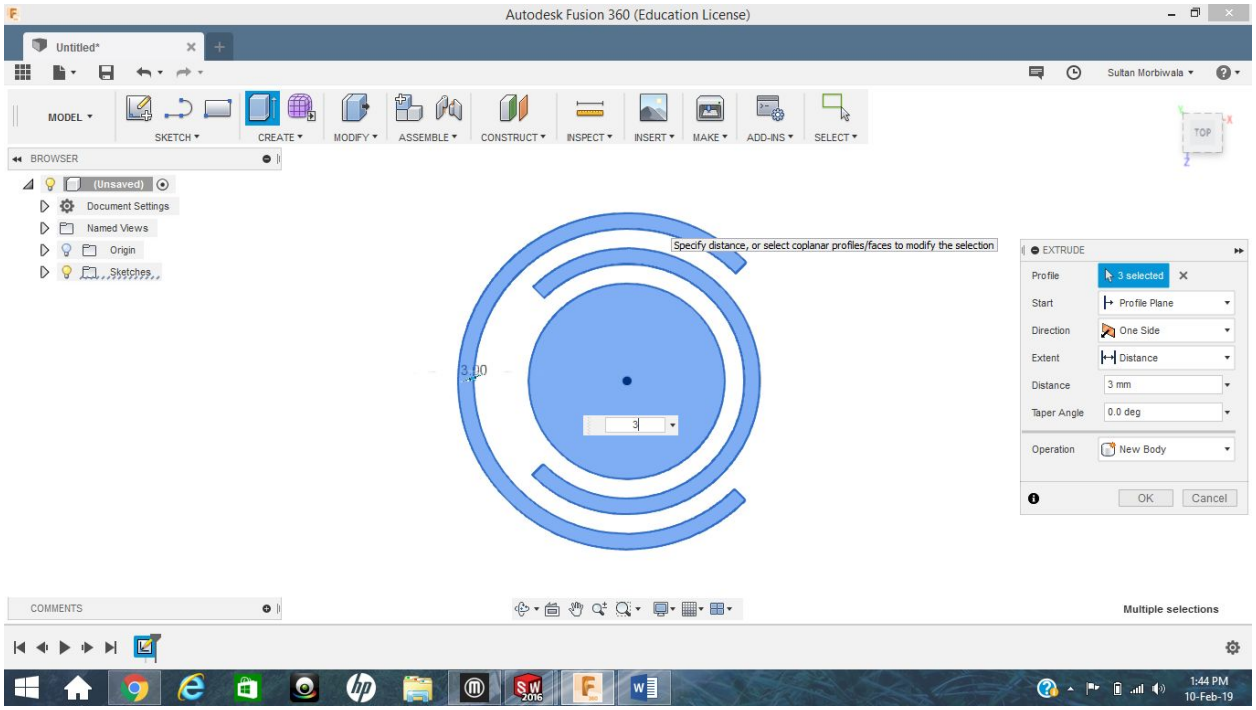




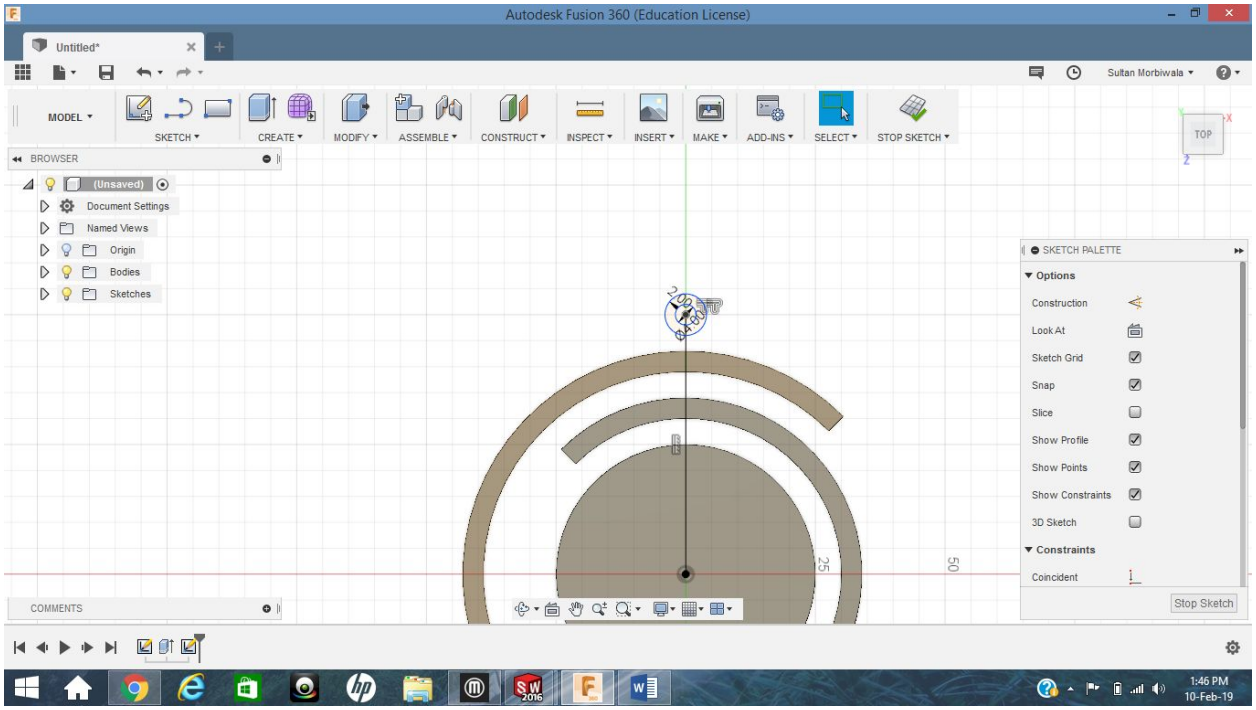


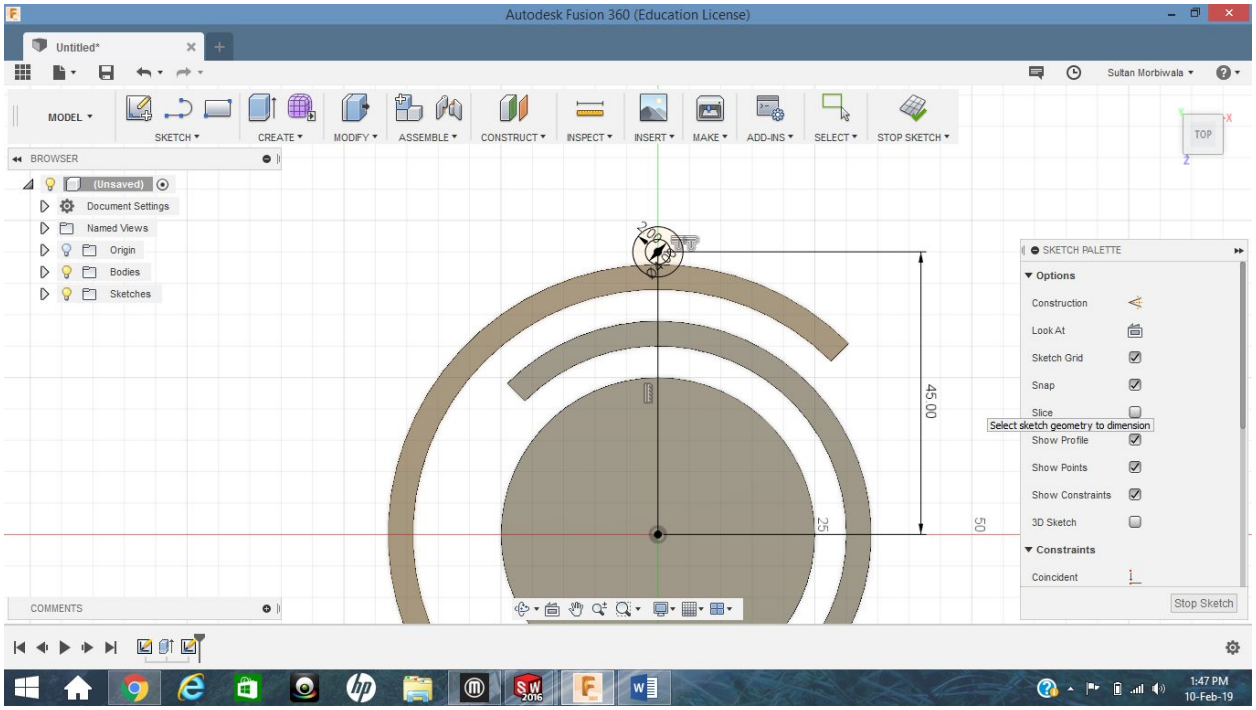
Step 4:- Use Extrude to make a solid surface with 3mm thickness

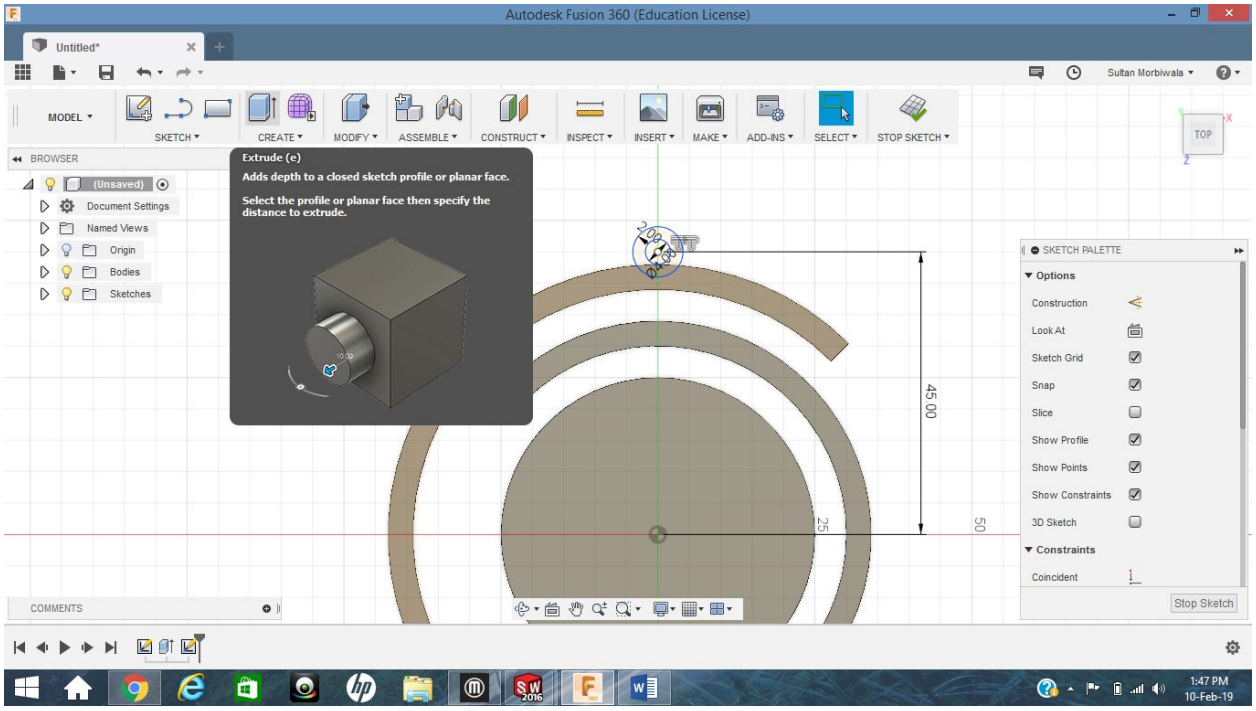




Step 5:- Draw a circle of 4mm which is 45mm away from center. Offset it by 2mm. Make extrude of 3mm below to join with the solid body.




Autodesk Fusion 360 (Education License)
Untitled*
Sultan Morbiwala
MODEL
SKETCH
CREATE
MODIFY
ASSEMBLE
CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
STOP SKETCH
TOP
BROWSER
(Unsaved)
Document Settings
Named Views
Origin
Bodies
Sketches
SKETCH PALETTE
Options
Construction
Look At
Sketch Grid
Snap
Slice
Show Profile
Show Points
Show Constraints
3D Sketch
Constraints
Coincident
Select sketch geometry to dimension
Stop Sketch
COMMENTS
45.00
25
50
1:47 PM
10-Feb-19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Sultan Morbiwala
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CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
STOP SKETCH
TOP
BROWSER
(Unsaved)
Document Settings
Named Views
Origin
Bodies
Sketches
Extrude (e)
Adds depth to a closed sketch profile or planar face.
Select the profile or planar face then specify the distance to extrude.
SKETCH PALETTE
Options
Construction
Look At
Sketch Grid
Snap
Slice
Show Profile
Show Points
Show Constraints
3D Sketch
Constraints
Coincident
Stop Sketch
COMMENTS
45.00
25
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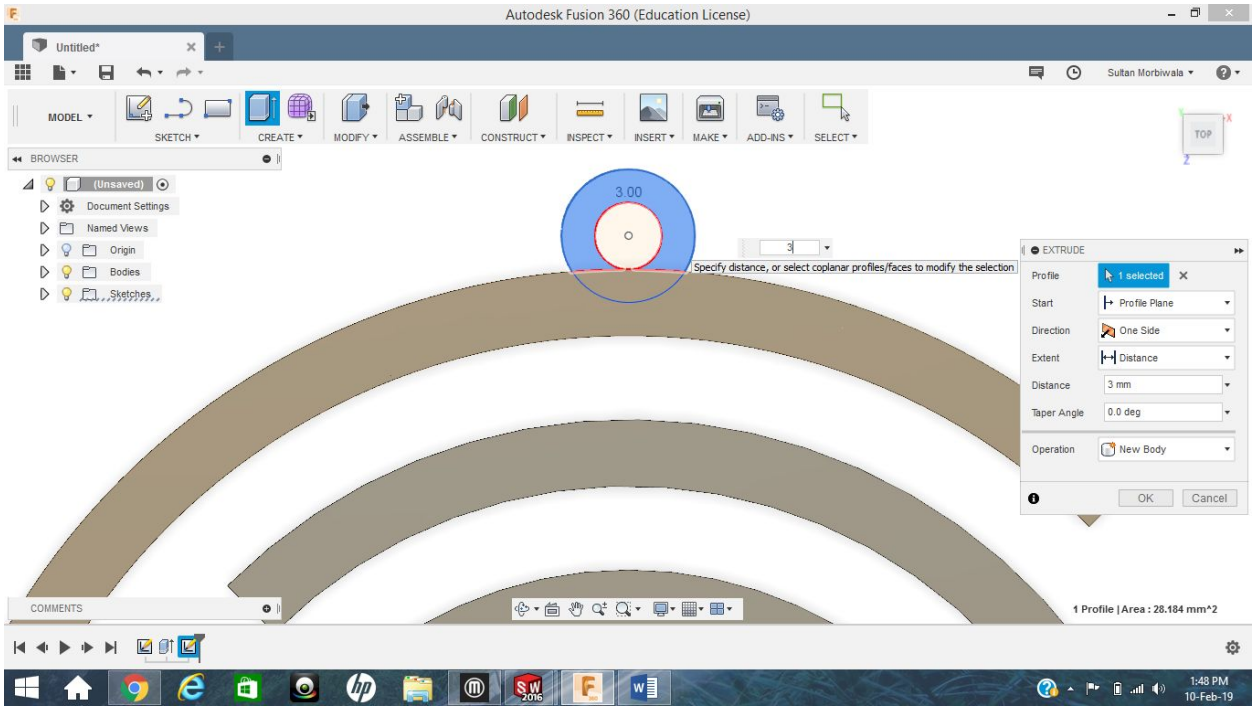
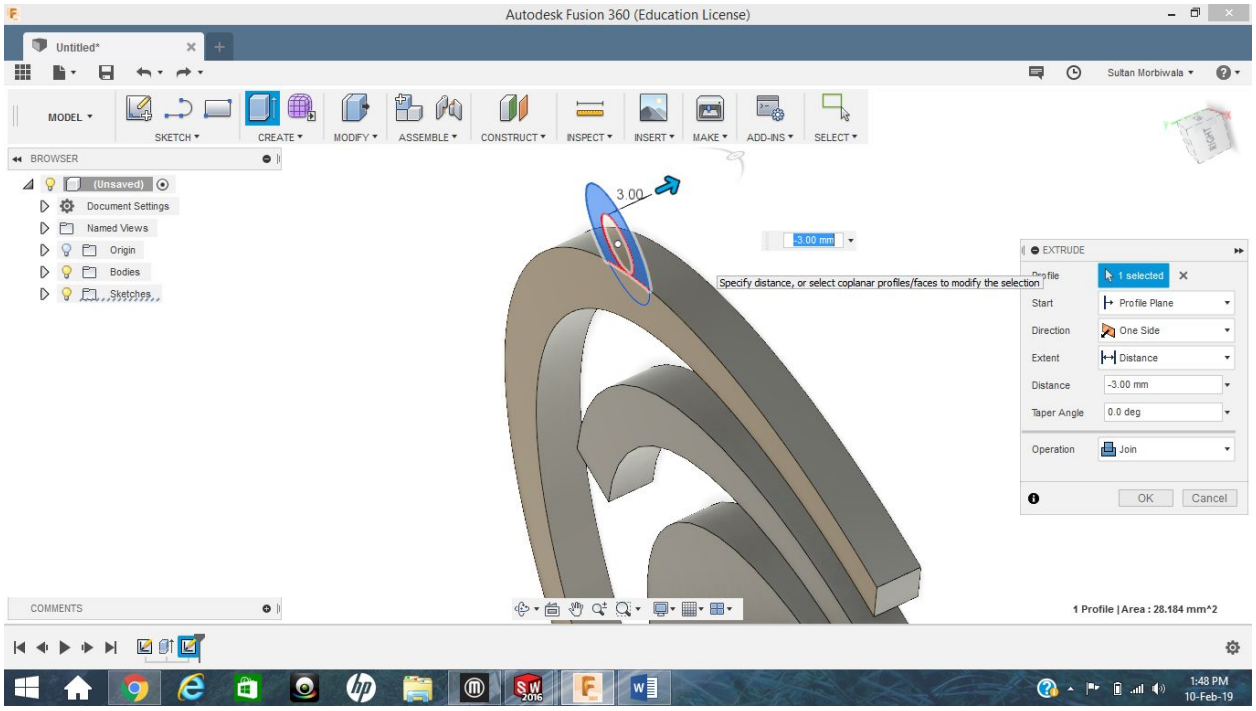
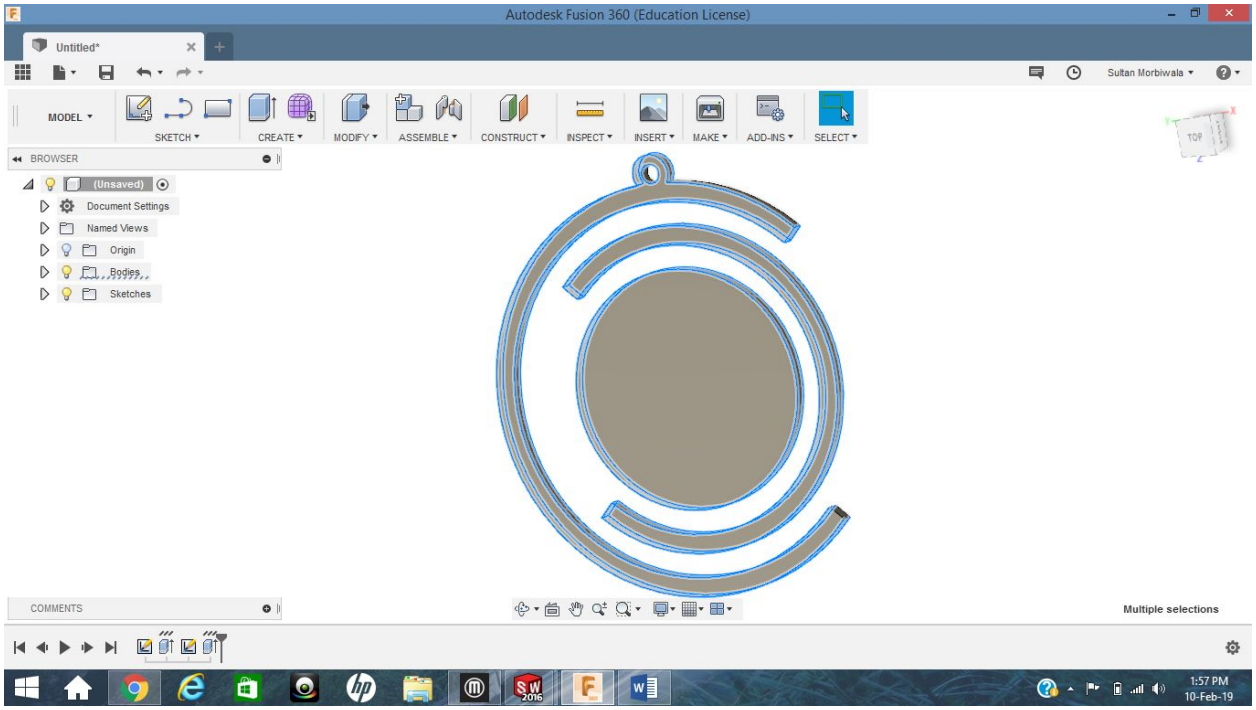
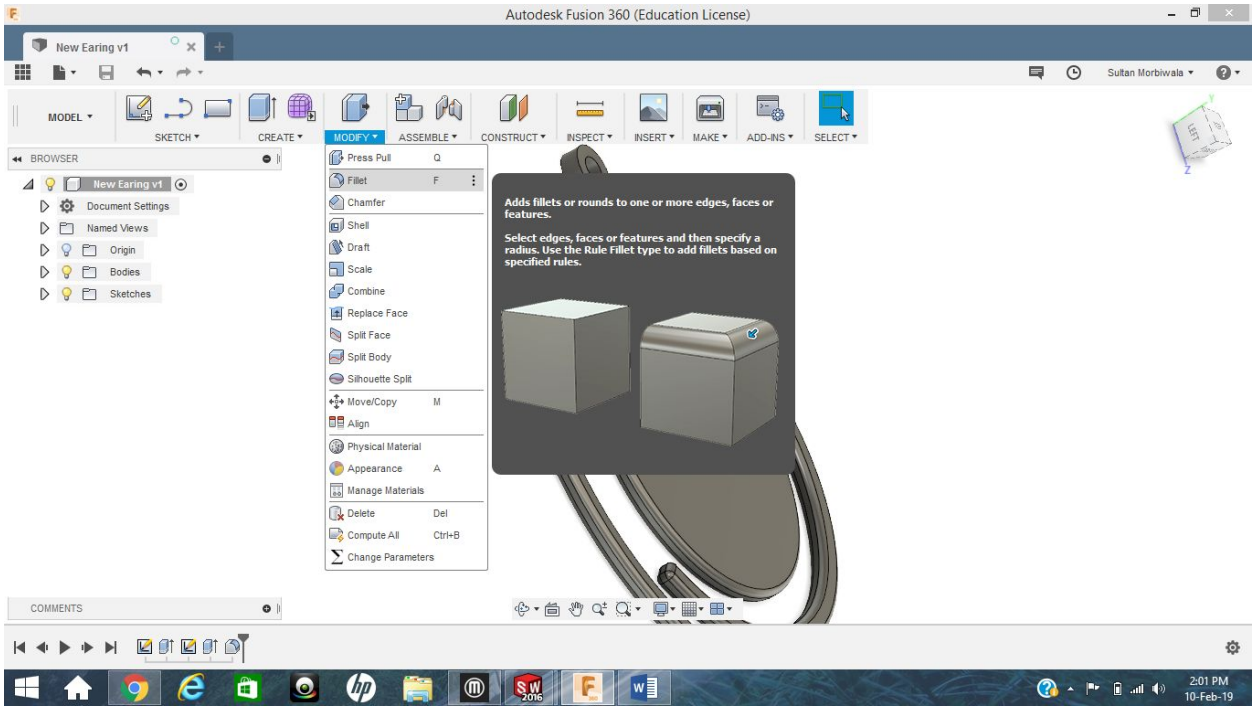
1:47 PM
10-Feb-19

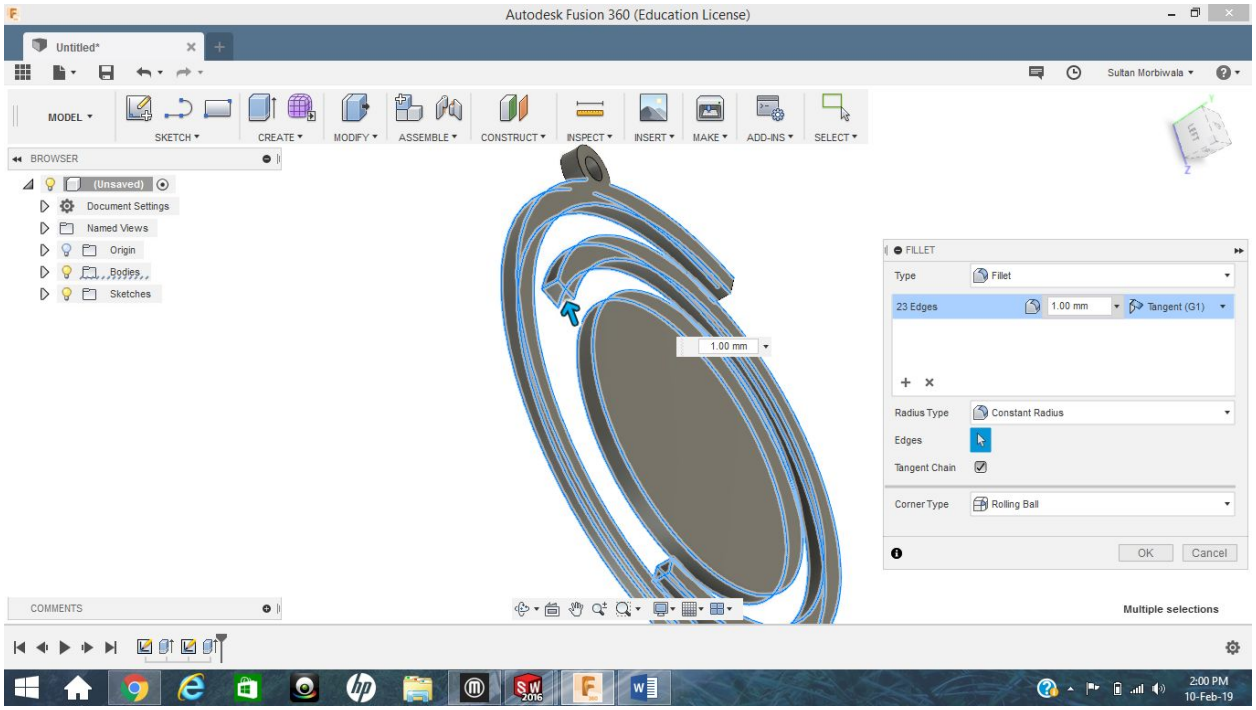

Autodesk Fusion 360 (Education License)
Untitled*
Sultan Morbiwala
MODEL
SKETCH
CREATE
MODIFY
ASSEMBLE
CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
BROWSER
(Unsaved)
Document Settings
Named Views
Origin
Bodies
Sketches
3.00
Specify distance, or select coplanar profiles/faces to modify the selection
EXTRUDE
Profile
1 selected
Start
Profile Plane
Direction
One Side
Extent
Distance
Distance
3 mm
Taper Angle
0.0 deg
Operation
New Body
OK
Cancel
COMMENTS
1 Profile | Area : 28.184 mm^2
1:48 PM
10-Feb-19


Autodesk Fusion 360 (Education License)
Untitled*
Sultan Morbiwala
MODEL
SKETCH
CREATE
MODIFY
ASSEMBLE
CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
BROWSER
(Unsaved)
Document Settings
Named Views
Origin
Bodies
Sketches
3.00
-3.00 mm
Specify distance, or select coplanar profiles/faces to modify the selection
EXTRUDE
Profile
1 selected
Start
Profile Plane
Direction
One Side
Extent
Distance
Distance
-3.00 mm
Taper Angle
0.0 deg
Operation
Join
OK
Cancel
COMMENTS
1 Profile | Area : 28.184 mm^2
1:48 PM
10-Feb-19

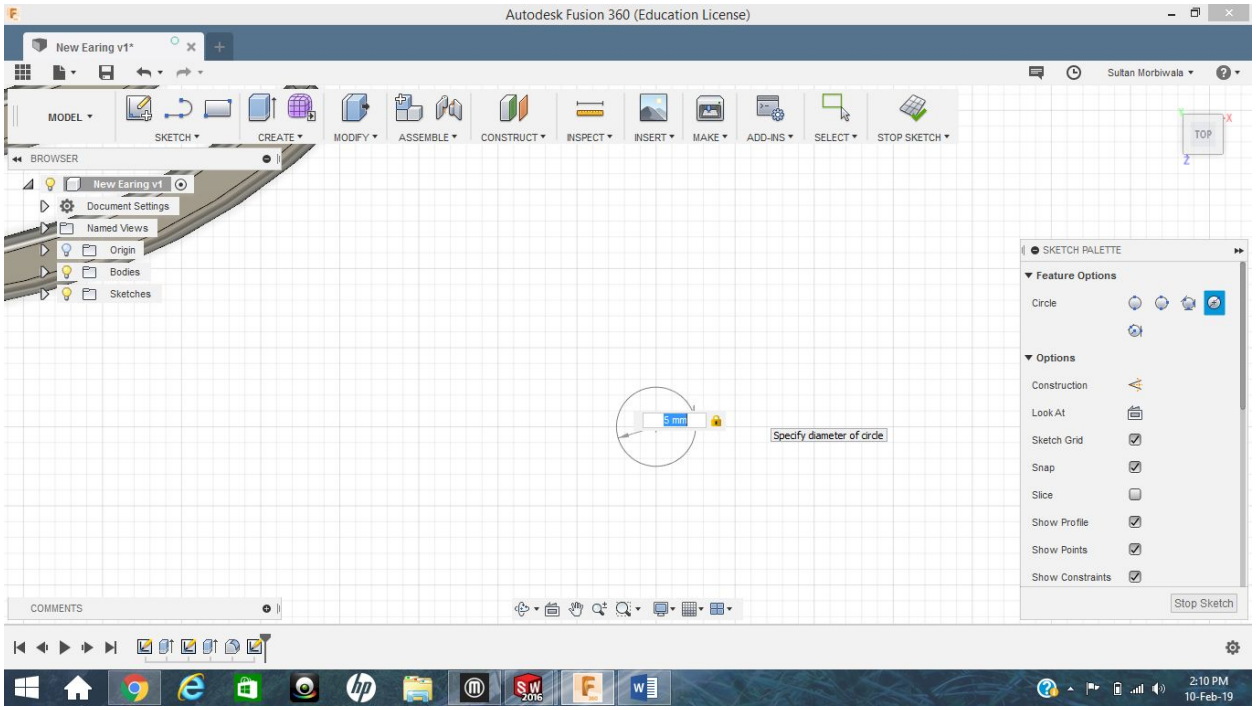


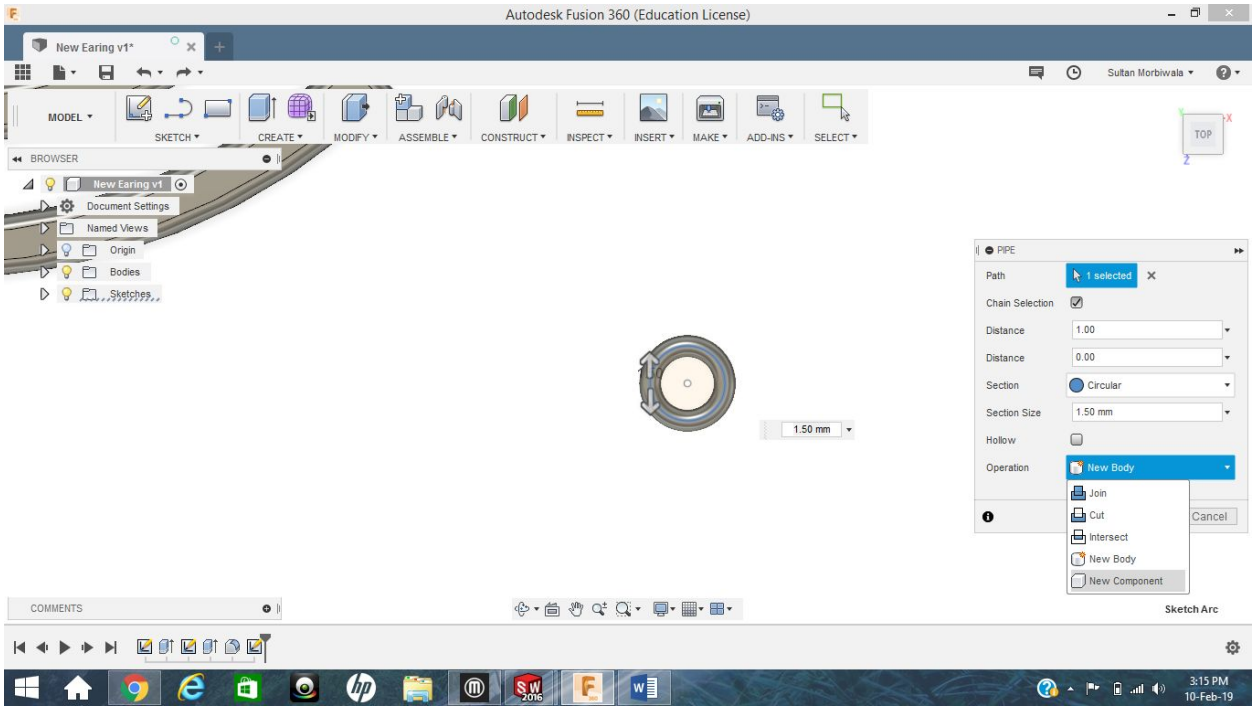
Step 6:- Select all the edges first and click on fillet feature. Fillet all sides by 1mm.



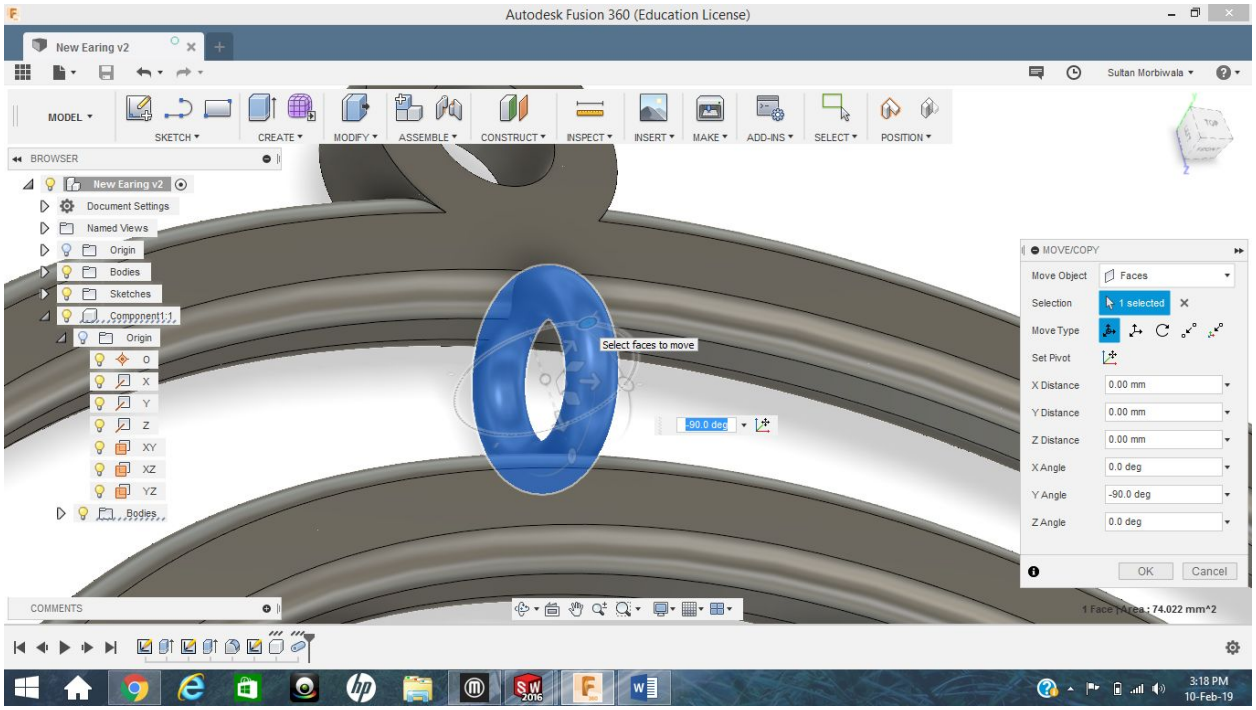


Step 7:- Draw a circle of 5mm diameter. Then use the pipe function to convert it into a ring with 1.5mm thickness.

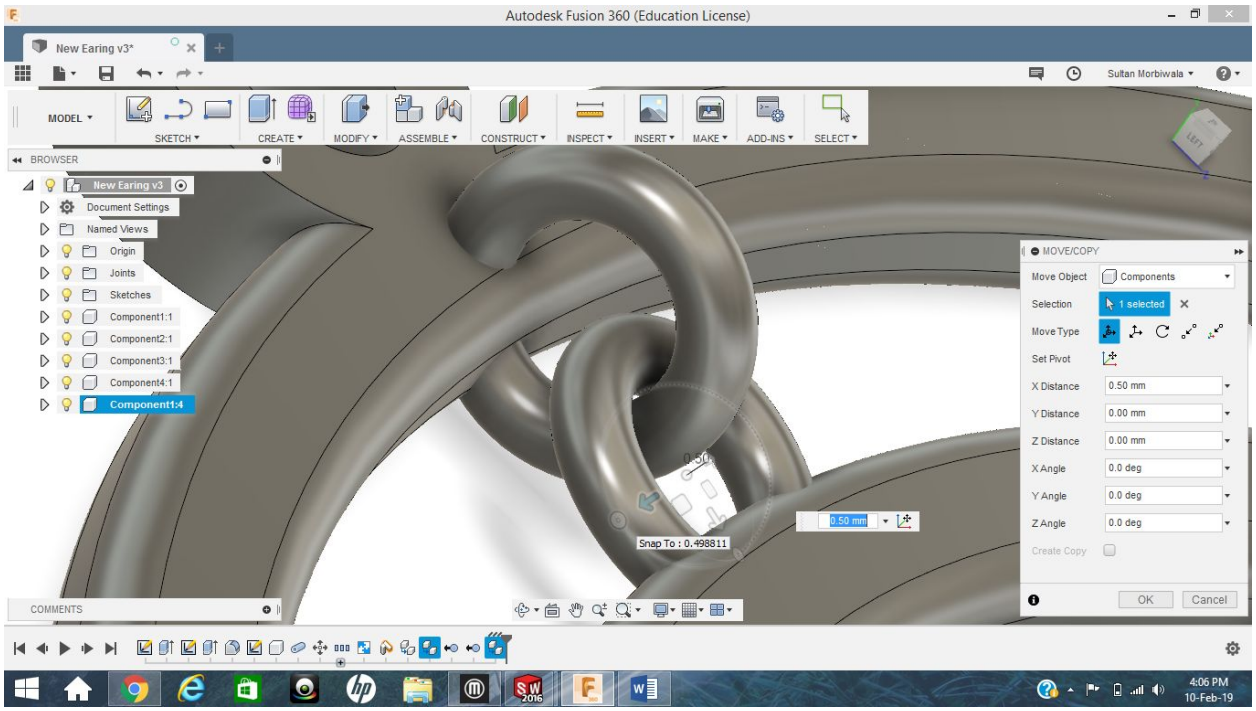




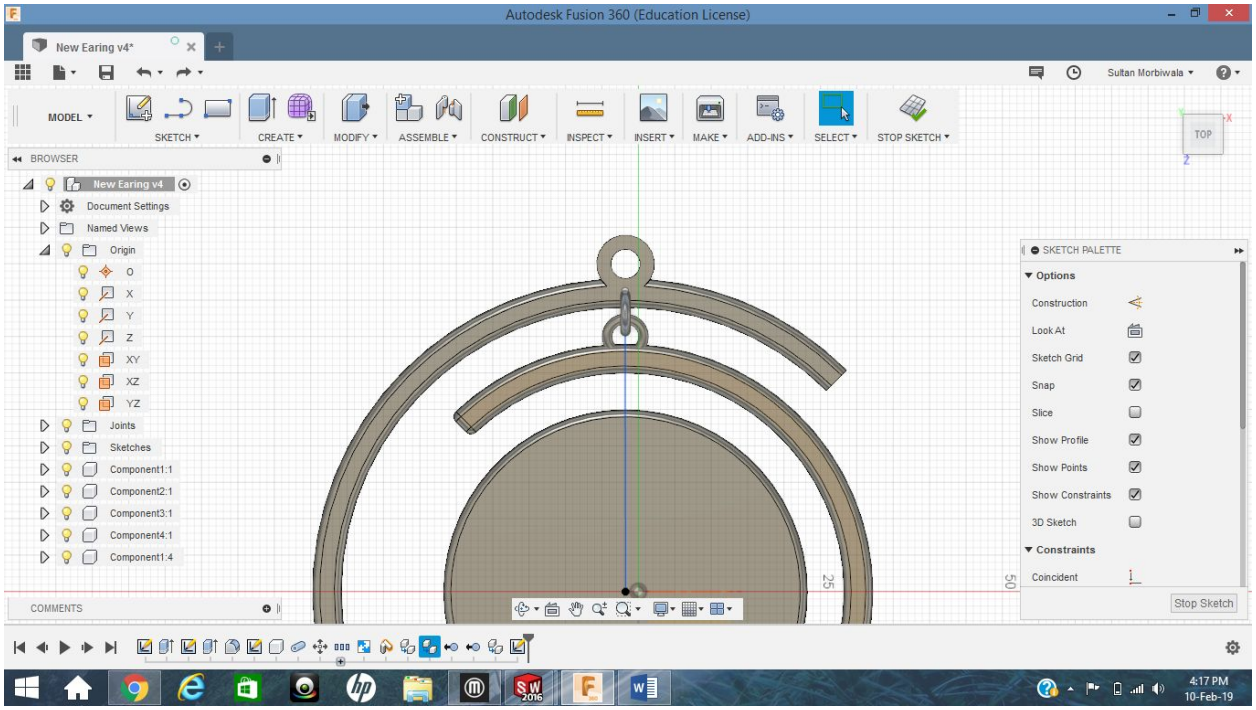
Step 8:- right click on the component and select move. Move the component at the position below.



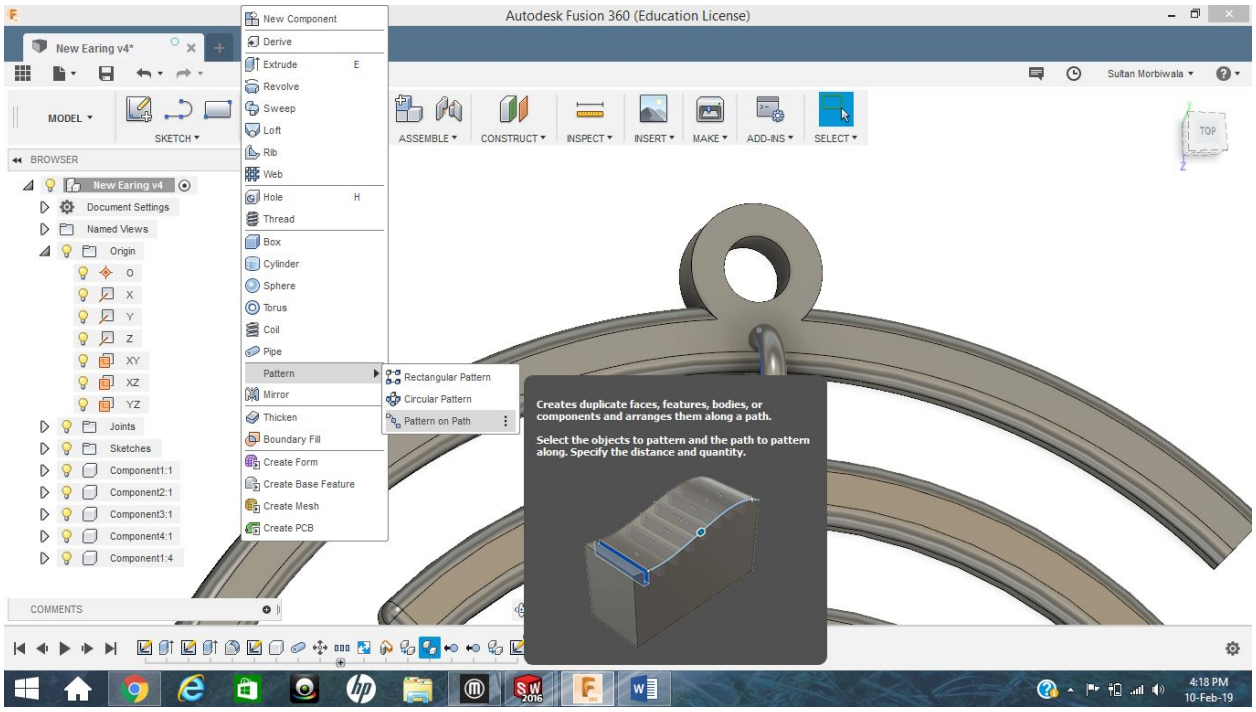
Step 8:- Create a second ring in the same way and position it like below.

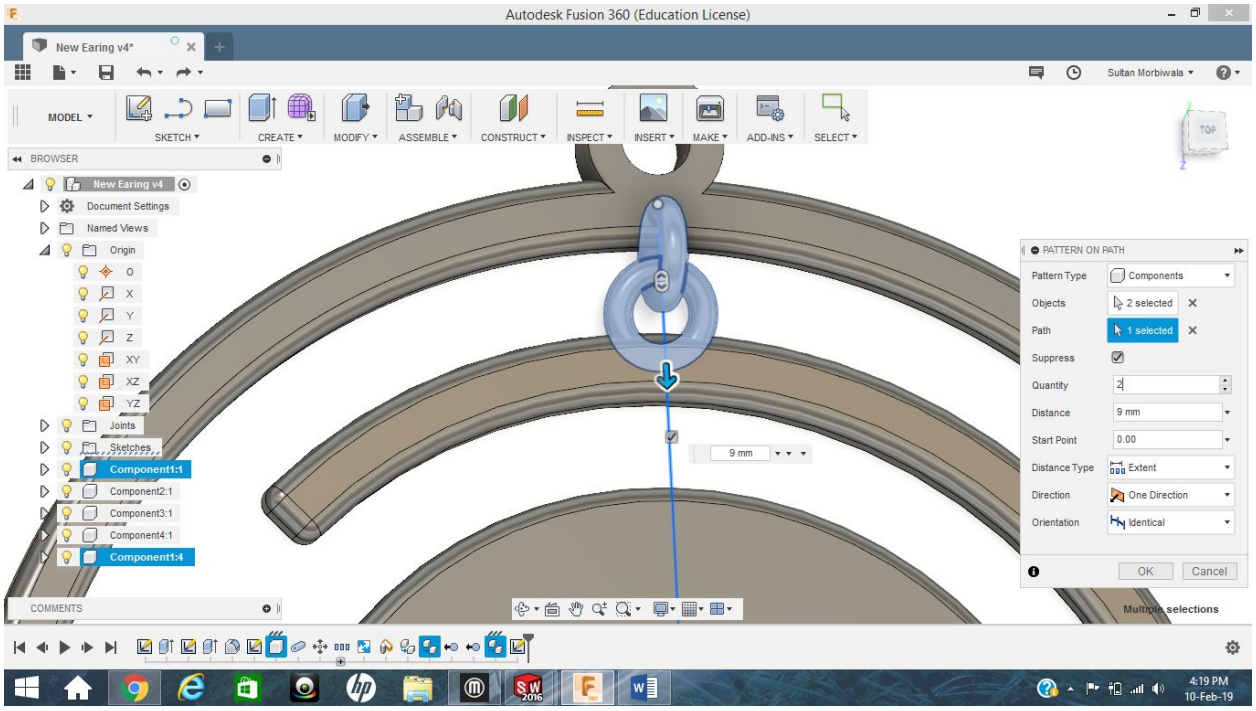


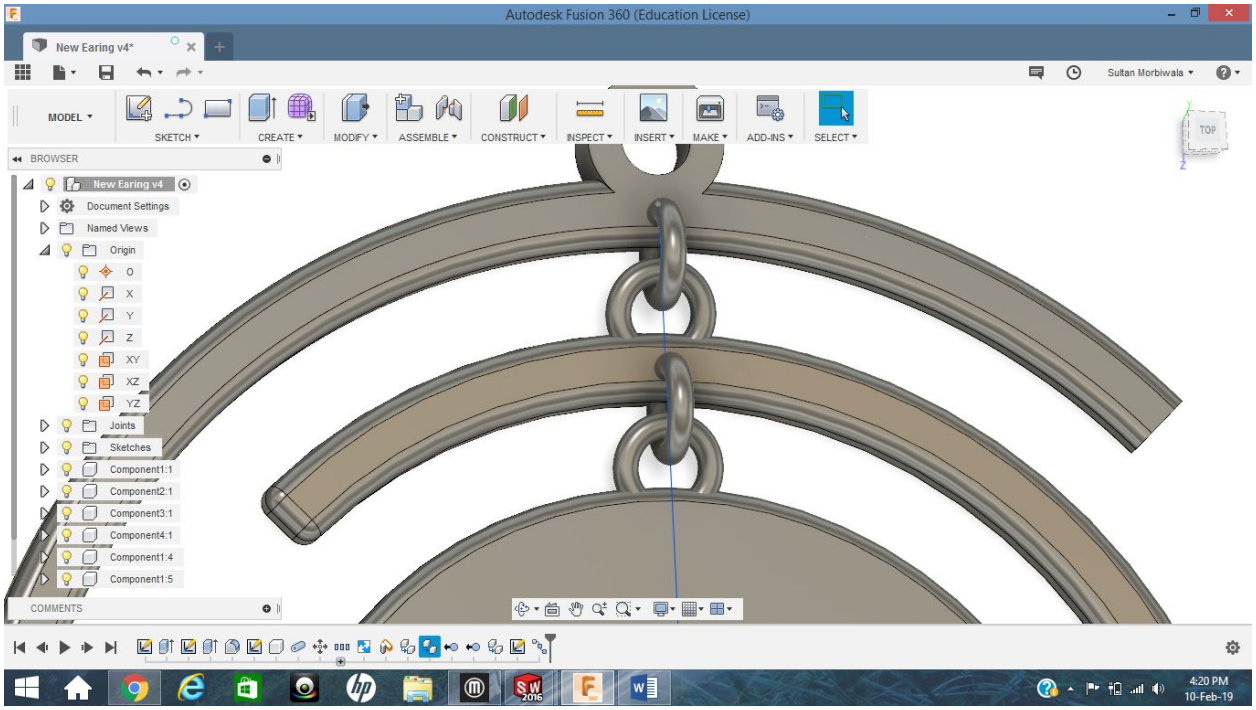
Step 9:- Draw a straight line on the surface which passes through the ring



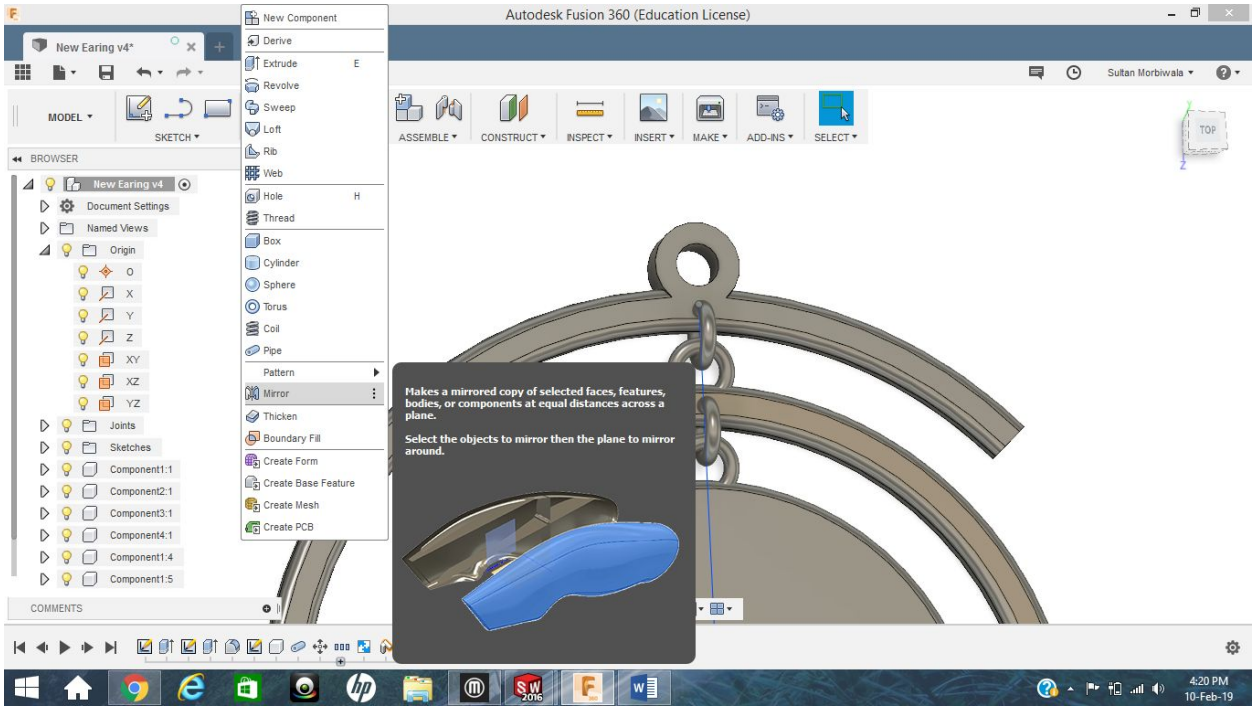
Step 10:- Using ‘pattern in parts’ feature, select 2 rings with settings below.

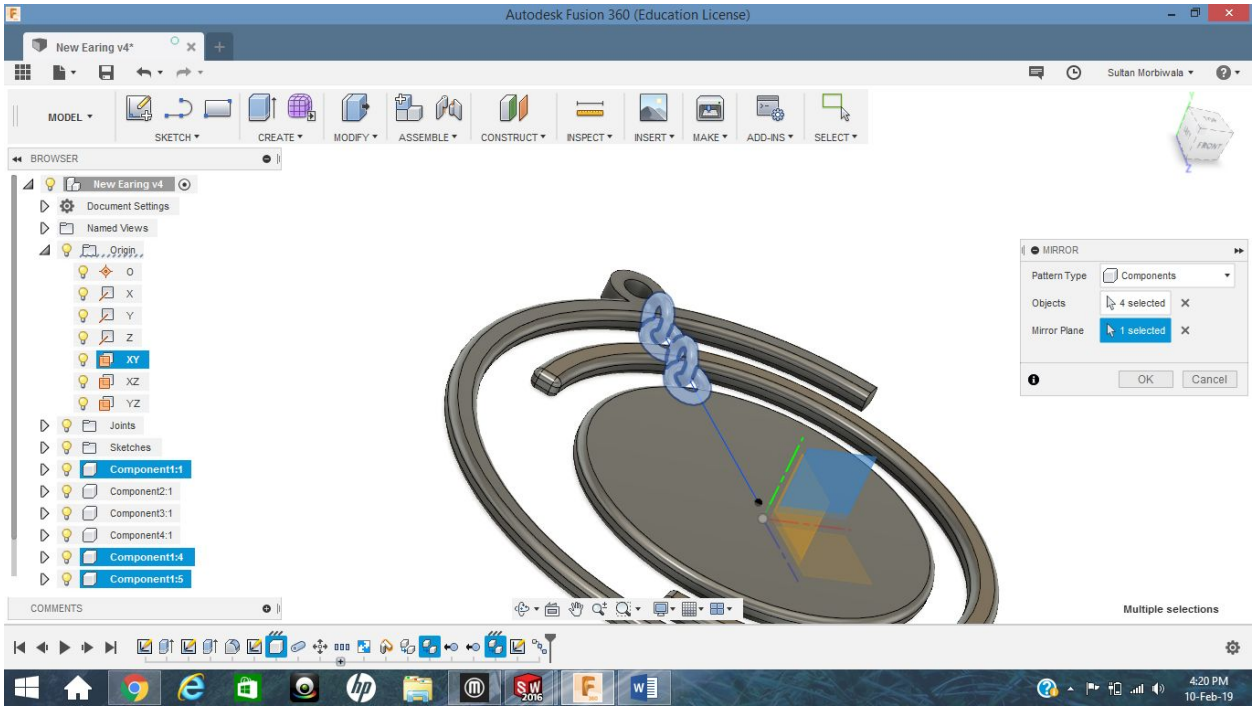






Step 11:- Use ‘mirror’ feature to mirror all the rings on XY plane




Autodesk Fusion 360 (Education License)
New Earing v4*
Sultan Morbiwala
MODEL
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ASSEMBLE
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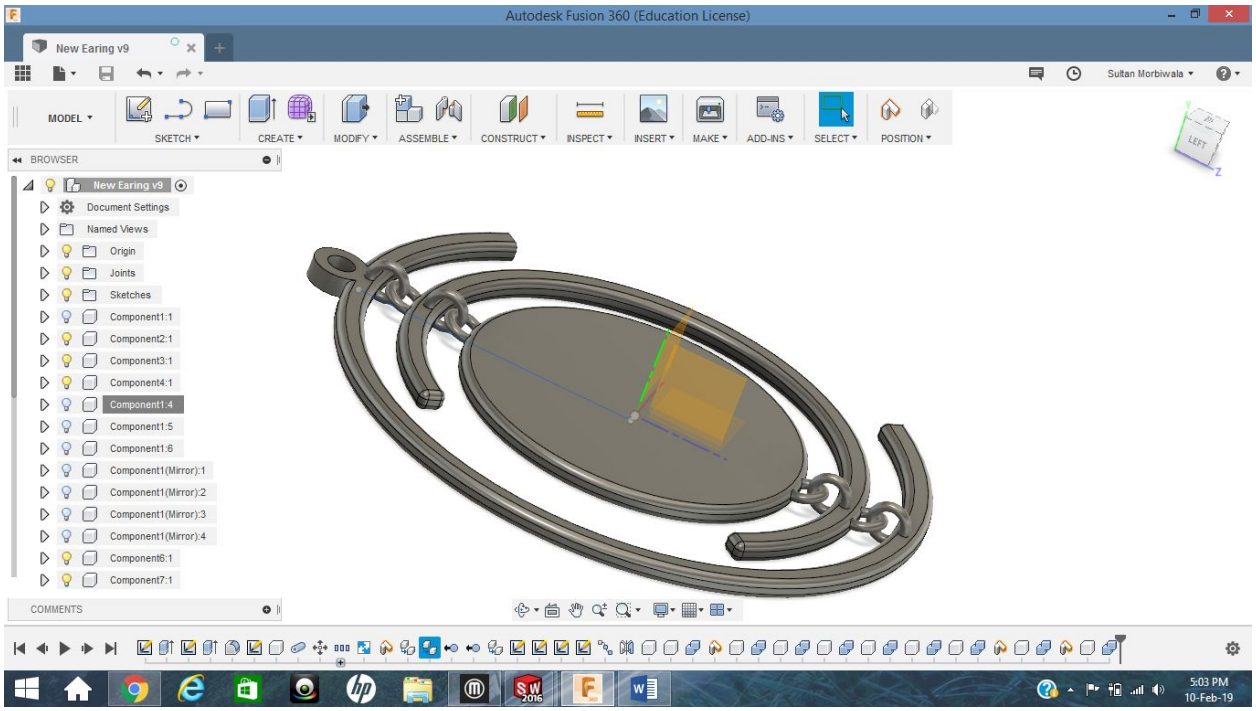
INSPECT
INSERT
MAKE
ADD-INS
SELECT
BROWSER
New Earing v4
Document Settings
Named Views
Origin
XY
XZ
YZ
Joints
Sketches
Component1:1
Component2:1
Component3:1
Component4:1
Component1:4
Component1:5
MIRROR
Pattern Type
Components
Objects
4 selected
Mirror Plane
1 selected
OK
Cancel
COMMENTS
Multiple selections
4:20 PM
10-Feb-19


Autodesk Fusion 360 (Education License)
New Earing v9
Sultan Morbiwala
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CONSTRUCT
INSPECT
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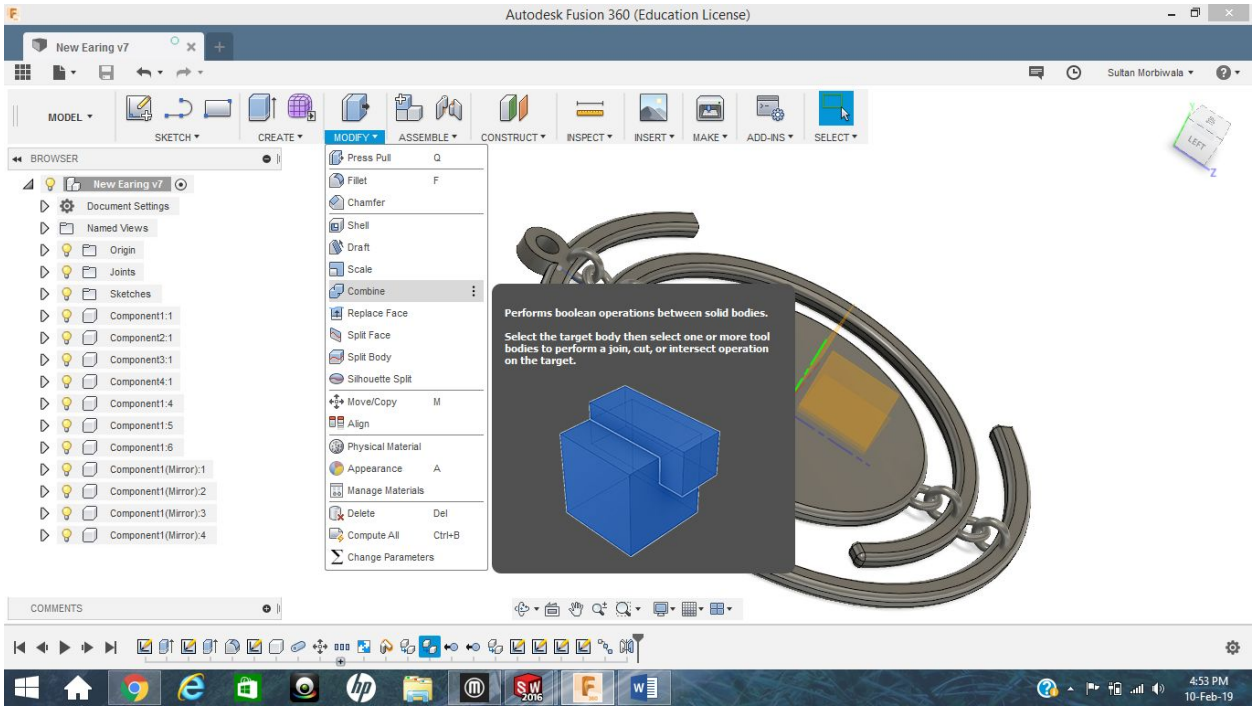
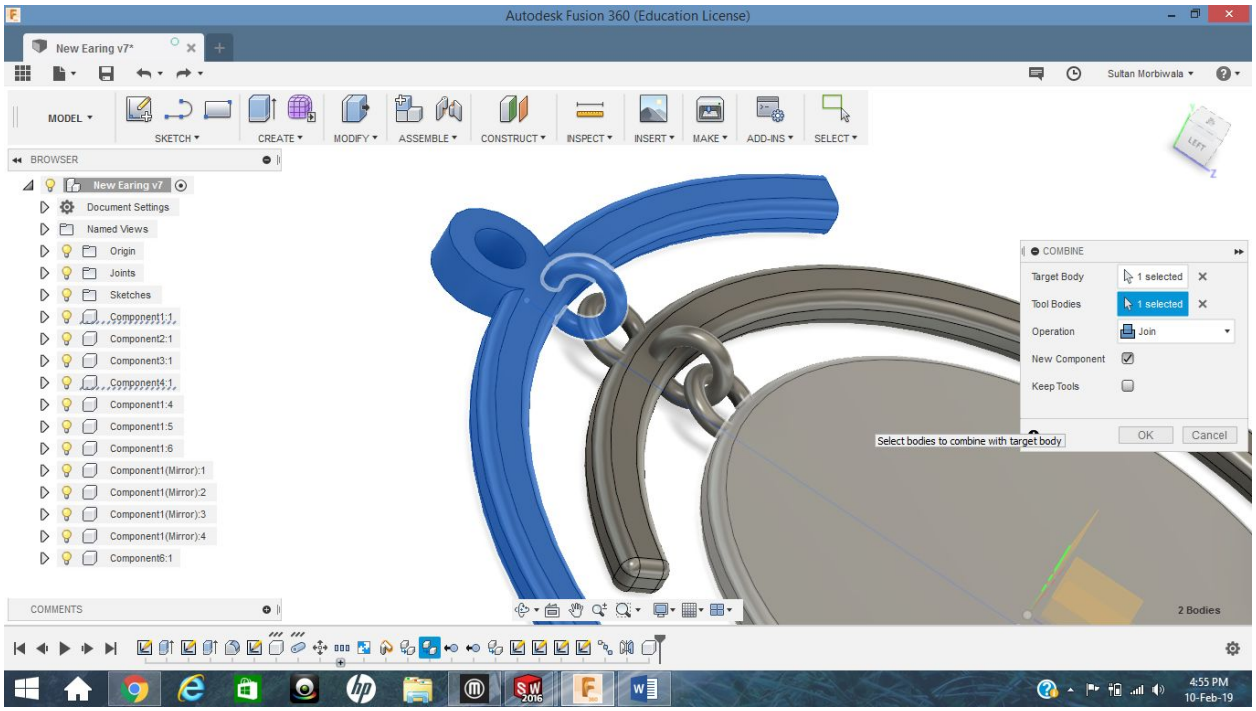
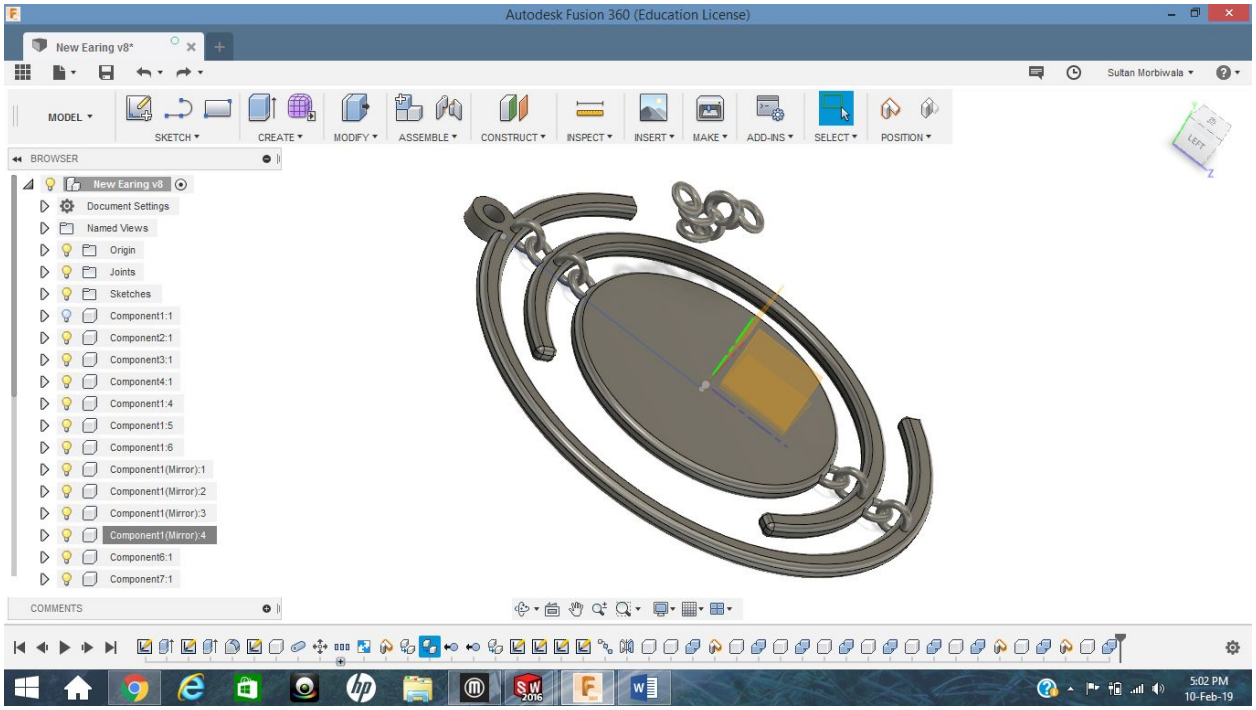
MAKE
ADD-INS
SELECT
POSITION
BROWSER
New Earing v9
Document Settings
Named Views
Origin
Joints
Sketches
Component1:1
Component2:1
Component3:1
Component4:1
Component1:4
Component1:5
Component1:6
Component1(Mirror):1
Component1(Mirror):2
Component1(Mirror):3
Component1(Mirror):4
Component6:1
Component7:1
COMMENTS
5:03 PM
10-Feb-19

Step 12:- Use combine feature to connect all the rings to the circular bodies. After combining everything, separate all the excess parts and hide it.



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New Earing v8*
Sultan Morbiwala
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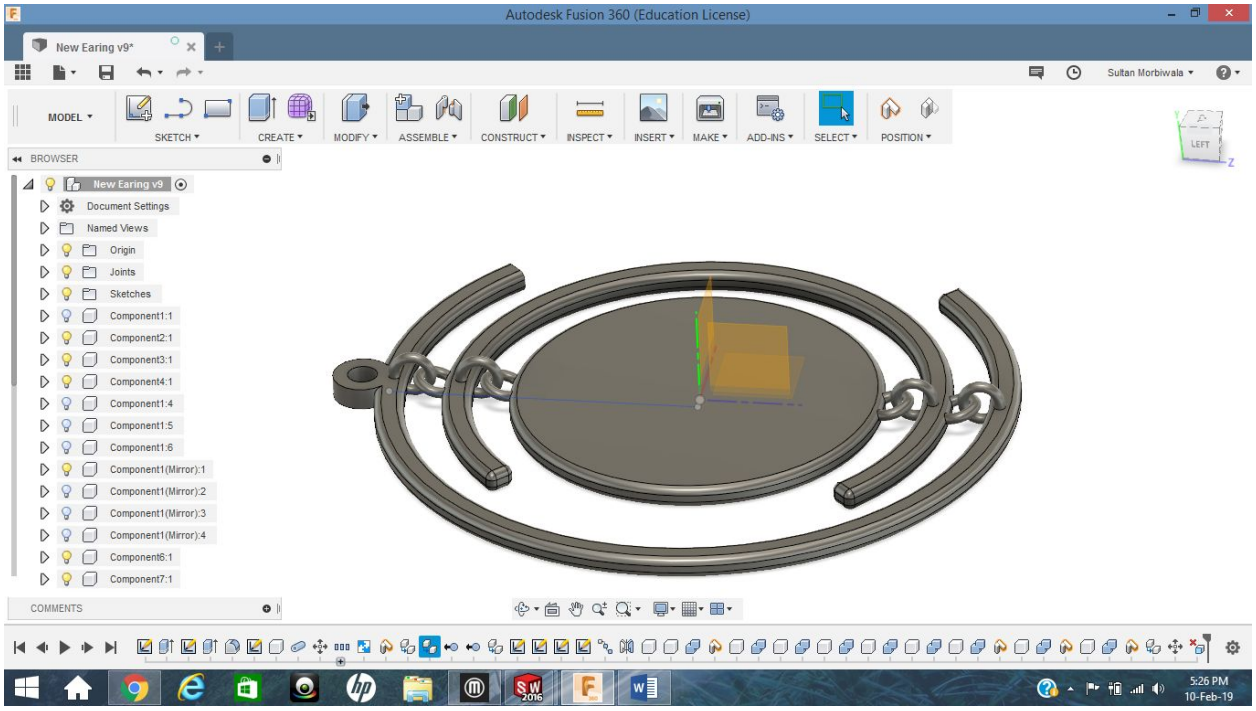
ASSEMBLE
CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
POSITION
BROWSER
New Earing v8
Document Settings
Named Views
Origin
Joints
Sketches
Component1:1
Component2:1
Component3:1
Component4:1
Component1:4
Component1:5
Component1:6
Component1(Mirror):1
Component1(Mirror):2
Component1(Mirror):3
Component1(Mirror):4
Component6:1
Component7:1
COMMENTS
LEFT
5:02 PM
10-Feb-19


Autodesk Fusion 360 (Education License)
New Earing v9*
Sultan Morbiwala
MODEL
SKETCH
CREATE
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ASSEMBLE
CONSTRUCT
INSPECT
INSERT
MAKE
ADD-INS
SELECT
POSITION
BROWSER
New Earing v9
Document Settings
Named Views
Origin
Joints
Sketches
Component1:1
Component2:1
Component3:1
Component4:1
Component1:4
Component1:5
Component1:6
Component1(Mirror):1
Component1(Mirror):2
Component1(Mirror):3
Component1(Mirror):4
Component6:1
Component7:1
COMMENTS
LEFT
5:26 PM
10-Feb-19