3D Printing

Lab Manual

For

Robotics Engineering

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DEPARTMENT OF ROBOTICS ENGINEERING
THE NEOTIA UNIVERSITY
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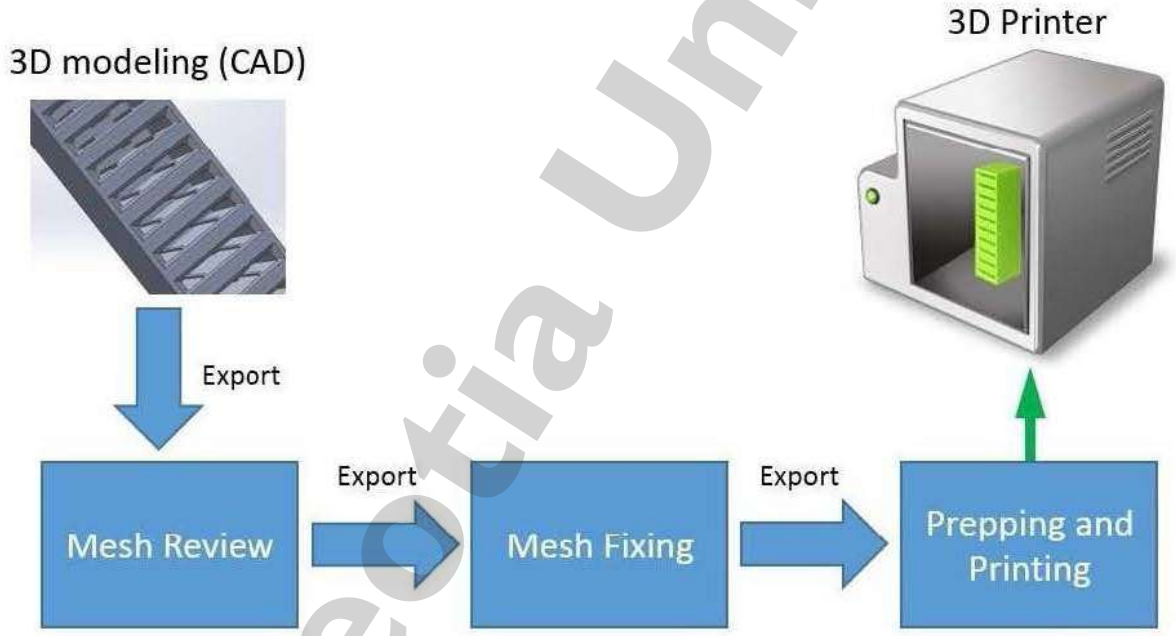
LIST OF EXPERIMENT

EXPERIMENT -1

Study of 3D Printing

Introduction: - 3D printing allows for rapid prototyping and onsite manufacturing of products. Initially done with plastic, 3D printing now uses new techniques with new materials, such as aluminum, bronze, and glass. Biomaterials are also being incorporated, such as 3D printing ear cartilage and liver tissue. As the 3D printing industry grows, 3D printing will become a big part of many engineering fields.

Flow layout of Pre 3D Printing



Components of 3D Printer: -

1. axes

Fixed Rods The three axes that the 3D printer utilizes are on the Cartesian coordinate system. The linear fixed rods are maintained at right angles to each other and each represents a coordinate axis.

Movement The timing belts and pulleys allow the movement of the hot end (or the print bed, depending on the type of 3D printer) along each axes according to the g-code (generated by slicing software). The stepper motors power this movement.

2. Extruder

Extrusion is the feeding of filament into the hot end of the 3D printer. This movement is also powered by a stepper motor.

Retraction This mechanism is the pulling of the melted filament from the hot end. This movement is primarily programmed through the g-code to prevent the formation of unwanted filament creating a

bridge between two areas. The bridging of unwanted filament is referred to as stringing or the formation of cobwebs.
Dual Extrusion Some models of 3D printers are equipped with dual extrusion capabilities. This allows for mixed material objects to be printed. Dual extrusion can be used to print out complex objects with a different colour material as the support, making it easy to differentiate between the object and the support.

3. Hot End

The hot end is heated to temperatures ranging from 160 C to 250 C, depending on the type of filament to be used. The hot end melts the filament and pushes the melted filament through the nozzle. The hot end needs to be thermally insulated from the other components of the 3D printer to prevent any damage.

4. Print Bed

Heated Print beds that are heated improve print quality of 3D printed objects. The heated bed is heated to the glass transition temperature of the filament being used. This allows the model layers to slightly melt and stick to the heated bed.
Non-Heated Print beds that are not heated require adhesion in the form of glue, tape, hairspray, etc. In the innovation lab, painters tape is frequently used for adhesion.

5. Filament

Filament is a consumable used by the 3D printer to print layers. Filament comes in a variety of materials and colors. Filament can be composed of metal, wood, clay, biomaterials, carbon fiber, etc.
i). **ABS**: - ABS is a thermoplastic that needs to be heated to temperatures from 210C to 250C. ABS can only be printed on a 3D printer with a heated bed, which prevents the cracking of the object. When ABS is heated, it emits a strong unpleasant odor. ABS requires a complete enclosure while printing.
ii). **PLA**: - PLA is a thermoplastic that needs to be heated to temperatures from 160C to 220C. PLA is also biodegradable and emits slight odors. PLA is most frequently used in the Innovation Lab on all 3D printers.

PVA PVA is a water soluble plastic that is frequently used for support in dual extrusion 3D printers. The printed object is left in water where the PVA support is dissolved and the finished object printed in the other filament remains.

Preparing your 3D Model in CAD Software: -

CAD software is used to create 3D models and designs. This software is available on our computers and the level of difficulty varies. With the exception of Sketch up Pro and the industry standard software mentioned, all of these programs are available on the innovation lab computers.
Solid works main idea is user to create drawing directly in 3D or solid form. From this solid user can assemble it directly on their workstation checking clashes and functionality of it. Creating drawing is pretty easy just drag and drop the solid to drawing block.

Preparing your 3D Model for print in Idea maker software:-

These are following step for 3D printing of model
Install the 3D print software idea maker
Check repair option in this software
Set the nozzle parameter and build tack temperature according to the printer guide.
Step:-1 Prepare the design Model using Designing Software(Solids Work,Autocad etc.)
Step:-2 Convert the designed Model file in Stl ,obj format.
Step:-3 Prepare the design model for printing Using Software Idea Maker and Ultimaker. Then set all parameter (nozzle temp., buildtak temp and support) and also repair your design using software option. Then after generate the file in gcode format

Step:-4 ON the 3D Printer and load the filament in nozzle and give the command print by using 3D Printing Machine.

Precaution of 3D Printer machine: -

These are some following precaution when you print the design in 3D Printer

Mechanical: Do not place limbs inside the build area while the nozzle is in motion. The printer nozzle moves in order to create the object.

High Temperature: Do not touch the printer nozzle – it is heated to a high temperature in order to melt the build material.

Always buy replacement parts from the manufacturer for safety related equipment

Choose an area that has adequate ventilation and exhaust capability

Safety Equipment: -

Safety Glasses

Gloves (recommended for postprocessing

Application of 3D Printer: -

Automotive

Marine

Aerospace

Advantages: -

Complex shapes

Freedom for design

Customize parts

Limitations: -

Time

Cost

Skill

Materials

Medical

Engineering

Architecture

Less waste

Fewer unsold products

Less transport

EXPERIMENT -2

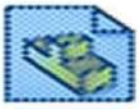
How to create a simple box

In this First exercise we will design a simple Box as seen in the figure below. This exercise will build the foundations of how to design 3D parts in Ultimaker Cura and introduce several fundamental features such as Sketching, Dimensioning and Extruding and printing on rise 3d machine. Let's begin. Check all Electrical connections.

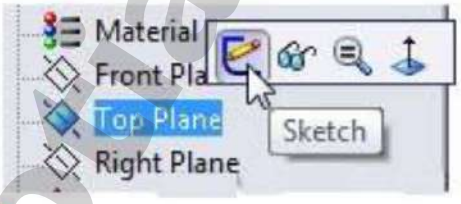
Step1. First we install 3d design software in your computer like Autocad etc

Step2. Open the Autocad software and click New option in software window

1. Click  , Option **New**



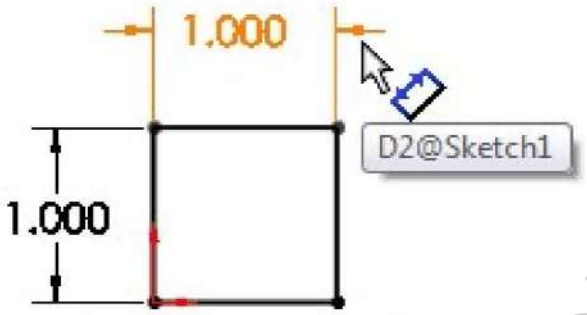
2. Click **Part** and **OK** .
3. Click on **Top Plane** and click **Sketch**.



4. Click  , sketch a rectangle start from origin.

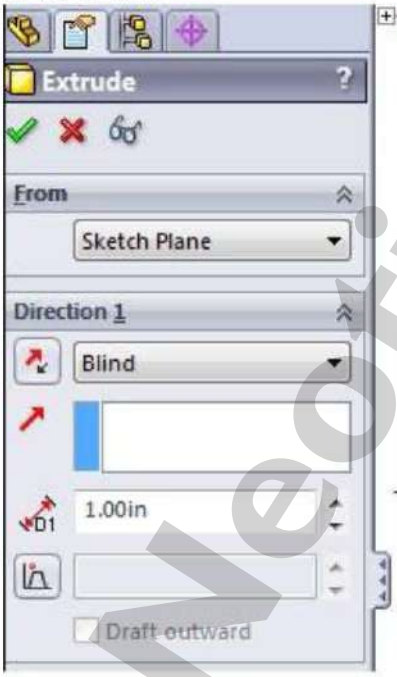
5. Click **Smart Dimension** 

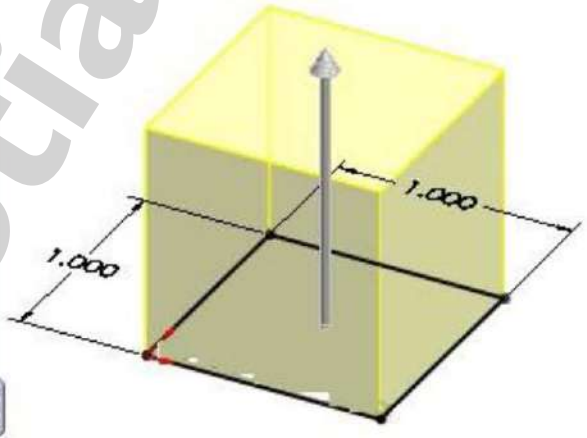
, click side edge and click top edge to dimension it as **1.0in x 1.0in**.



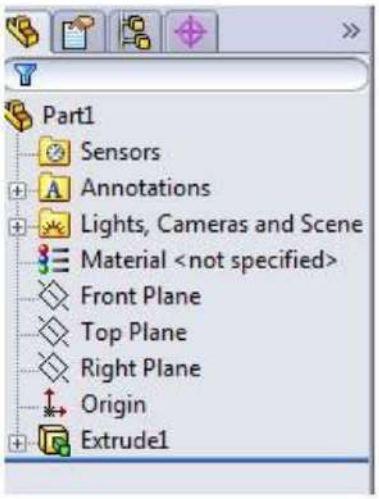
6. Click **Features>Extruded Boss/Base**

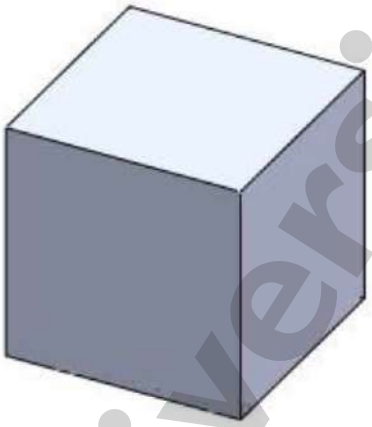
set **D1** as **1.0in**





and click ✔.



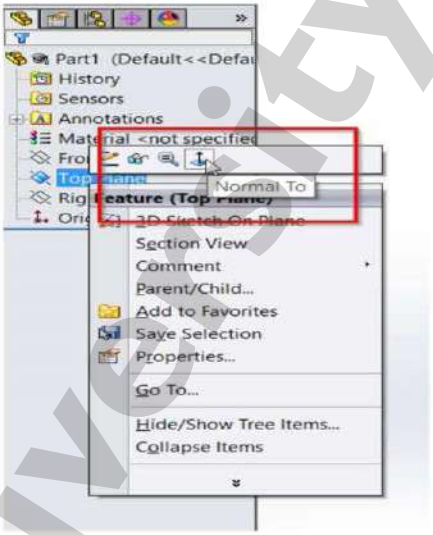


EXPERIMENT -3

To Design A Basic Helix Shape

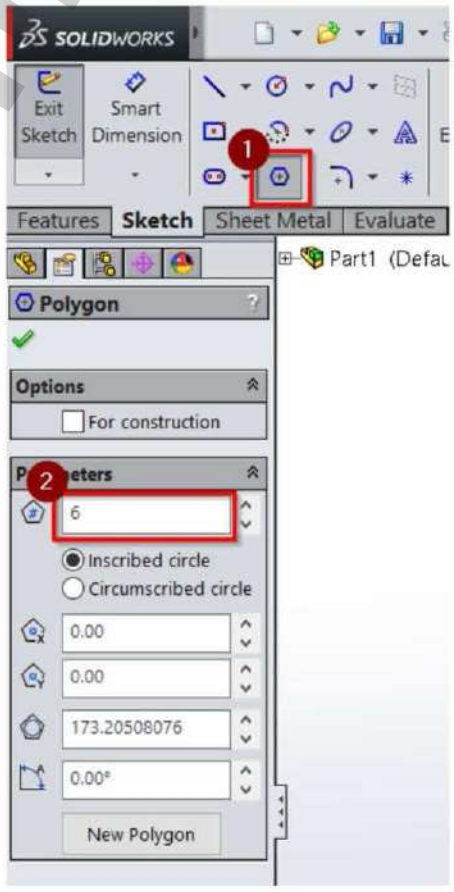
In this Second exercise we will design a simple hexagonal unthreaded nut as seen in the figure below. This exercise will build the foundations of how to design 3D parts in SolidWorks and introduce several fundamental features such as Sketching, Dimensioning and Extruding and printing on rise 3d machine. Let's begin. Check all Electrical connections.

Step1. First, start a new part by going to **New > Part > Ok**. Since we have already set up SolidWorks to use MMGS as default units, all our units should be in millimeters, grams and seconds. You should now be greeted with the main drafting interface of SolidWorks.



Step2. On the left-hand side, you will see the Feature Manager Design Tree. Right click on "Top Plane" and click the "Normal To" option. Now we are facing the Top Plane head-on.

Step 3. While the Top Plane is selected, switch to the Sketch tab at the top of the interface and select the "Sketch" tool in the top left corner. We are now creating a sketch on the Top Plane. Anything we sketch will appear on the Top Plane. First, we want to sketch the basic hexagonal outline of the nut.

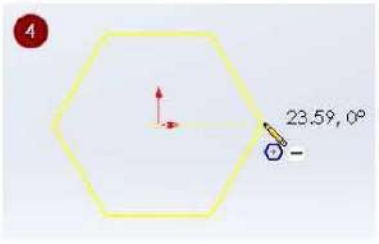


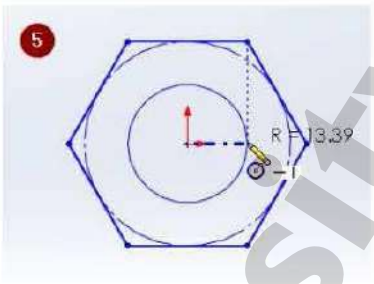
i. Click the "Polygon" tool in the Sketch tab. A list of parameters should appear in the left-hand panel.

ii. Set the number of sides to 6 (hex = 6). Other parameters such as center position, angle, etc. are also modifiable, but we will not adjust these.

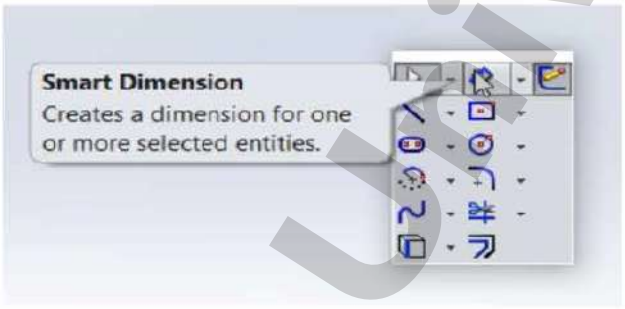
iii. Next,over the origin (the base of the two red arrows) until the cursor snaps onto the dot in the middle. Left click once to start the sketch. iv. Left click again at some point to the right of the first point. Press enter to finalize the sketch.

v. Now, select the "Circle" tool in the Sketch tab and sketch a circle starting at the origin, similar to how we sketched the hexagon.Press enter to confirm the circle sketch.





4. Now we will assign dimensions to our outline. Make sure we are still in sketch mode by looking at the top left corner and seeing that the Sketch button is depressed and now reads "Exit Sketch". Press the "S" key on the keyboard. In the small window, click the "Smart Dimension" tool.

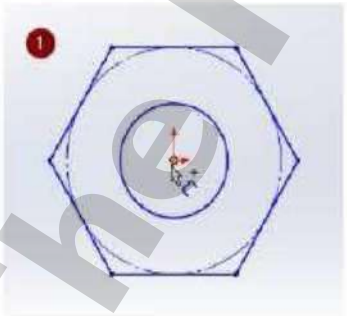


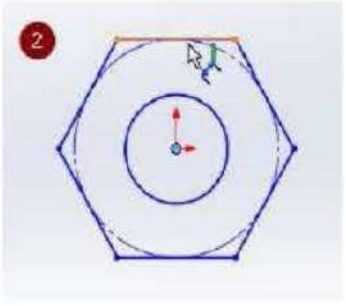
Now let's set the center-to-edge dimensions of the hexagon,

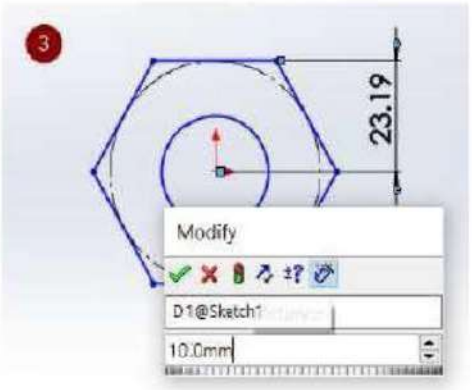
i. Left click the center

ii. Then left click the top side (be sure to select a side edge and not one of the points between two edges).

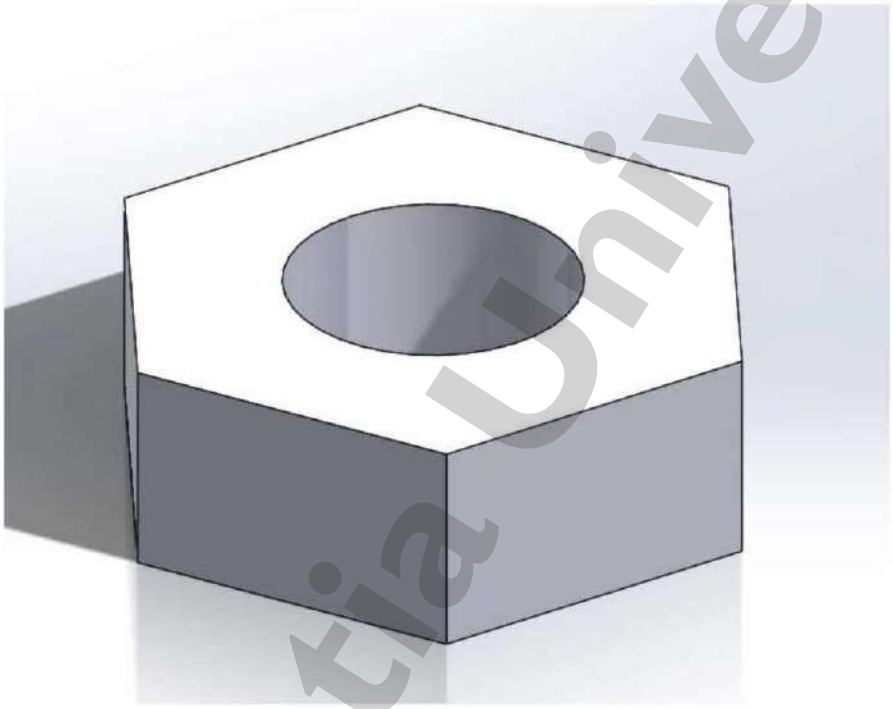
iii. A dimension measurement should pop up. Left click again anywhere on the screen and a small window will appear. Enter 10mm into this field and hit enter. The hexagon is now assigned a dimension of 10mm (center to edge). Double click the dimension label to change the assigned dimension at any time. Follow the same procedure to set the diameter of the circle to 10mm. Select Exit Sketch when finished. The sketch should now appear as follows:







5. Now that our sketch is assigned dimensions and finalized, we are ready to bring this shape into the 3rd dimension. Select our sketch by clicking "Sketch1" in the left panel. The sketch should turn light blue as it is selected. Next, switch to the Features tab at the top and click the "Extruded Boss/Base" tool. Immediately you should see the shape gain depth. Using the parameters on the left panel, give the nut a depth of 7.5mm. Hit enter to confirm the extrusion. The shape should now appear as follows:



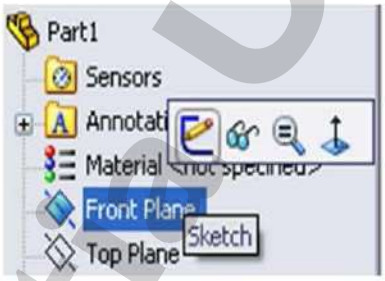
EXPERIMENT -4

How to create U bracket Sheetmetal

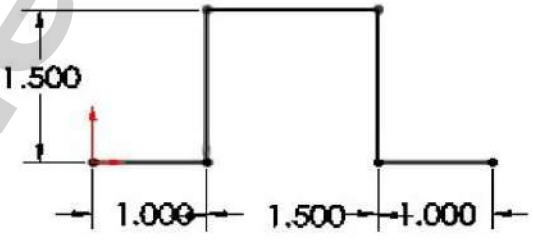
In this exercise, we will design a U Bracket Sheetmetal as seen in the figure below. This exercise will build the foundations of how to design 3D parts in SolidWorks and introduce several fundamental features such as Sketching, Dimensioning and Extruding and printing on rise 3d machine. Let's begin. Check all Electrical connections.

These are following step for making of U bracket sheetmetal.

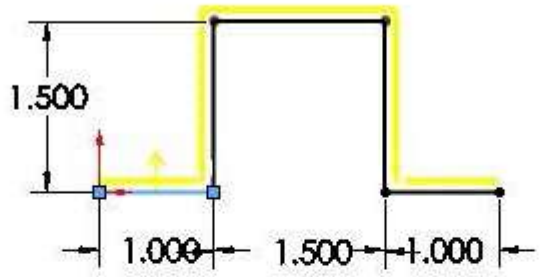
1. Click New. Click Part, OK.
2. Click Front Plane and click on Sketch.



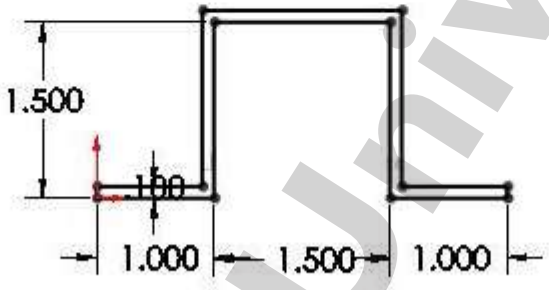
Use Line , sketch U shape. Dimension sketch with Smart Dimension as 1in x 1.5in x1in and 1.5in height.



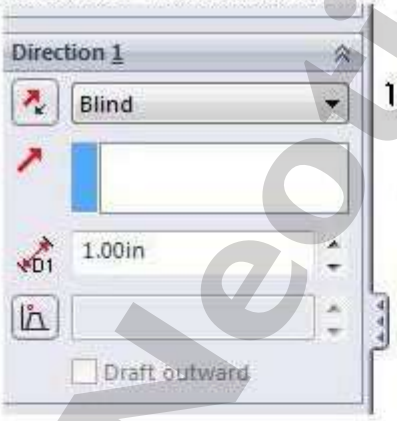
3. Click Offset Entities and click U sketch. Set offset distance as 0.1in, check Reverse box and OK.

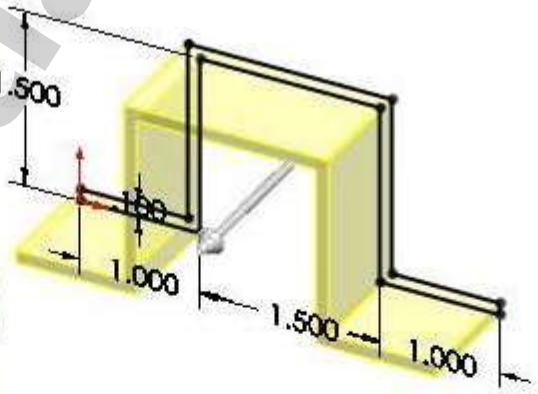


4. Use Line , sketch and connected open end of this sketch and make it close both end.



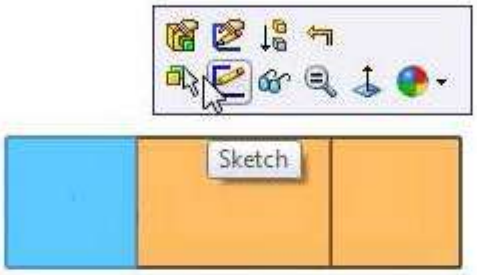
5. Click Features>Extruded Boss/Base set D1 to 1in and OK.



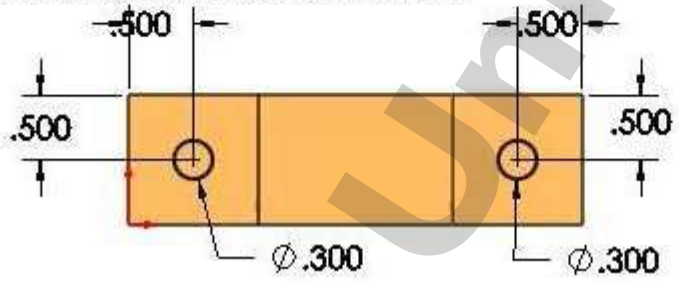


6. Click View>Bottom

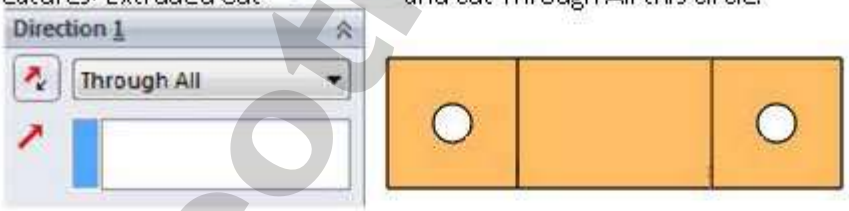
click on bottom face and click Sketch.



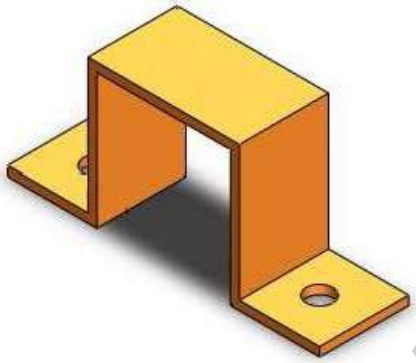
7. Click Circle and sketch 2 circle on bottom face each side. Use Smart Dimension to dimension this sketch as sketched below.



8. Click Features>Extruded Cut and cut Through All this circle.

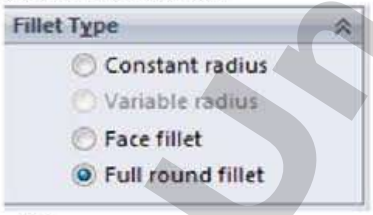


9. Click View>Isometric.

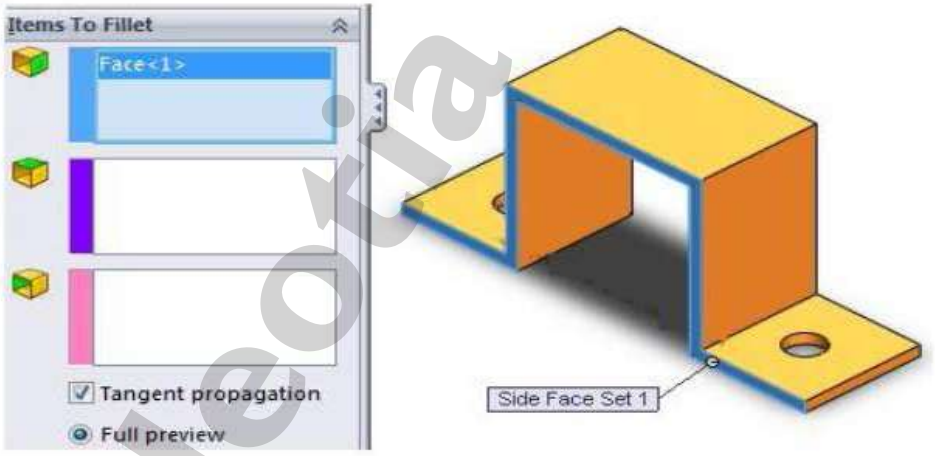


10. Click Fillet 

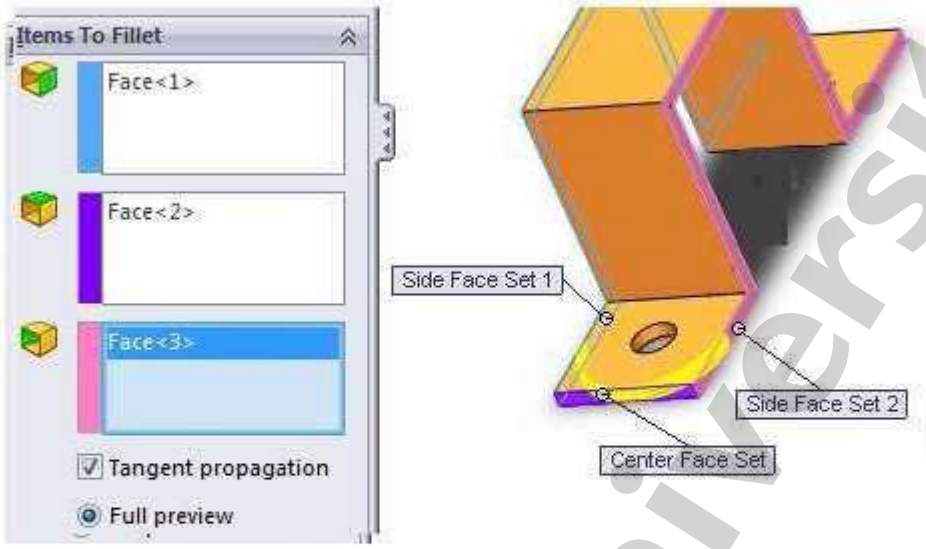
, check box Full round fillet.



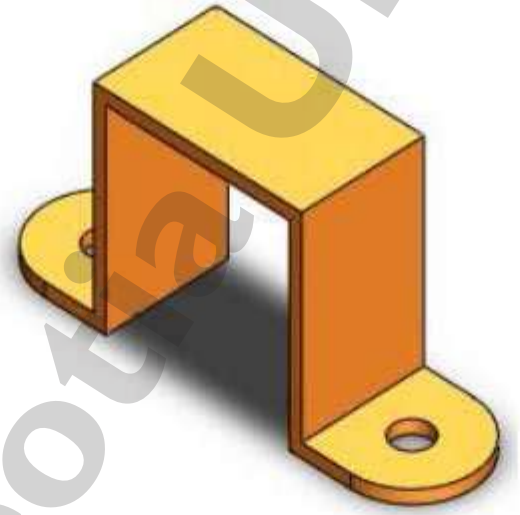
11. Click side left side face as Side Face 1.



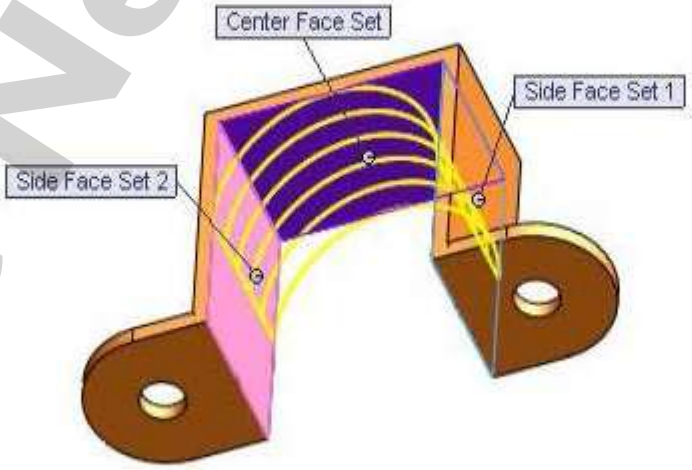
12. Click on pink box and click right side face as Side Face Set2 and OK.

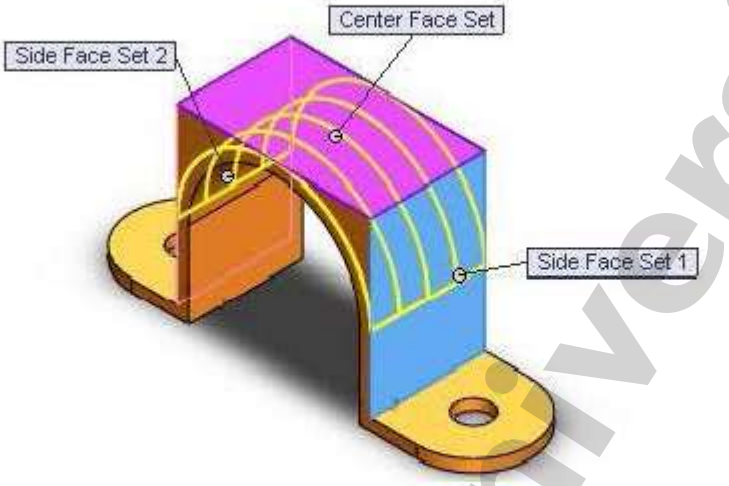


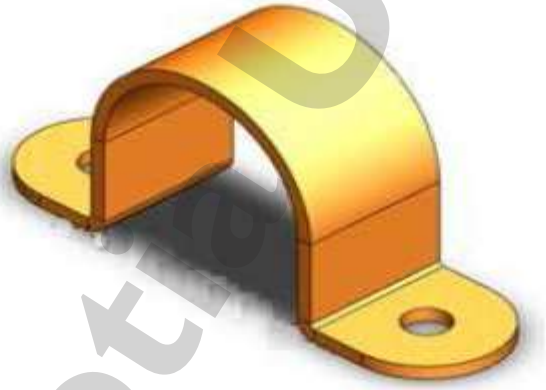
13. Repeat step 11 – 13 for the other side.



14. Repeat step 11 – 13 for inner face and outer face of U bracket.

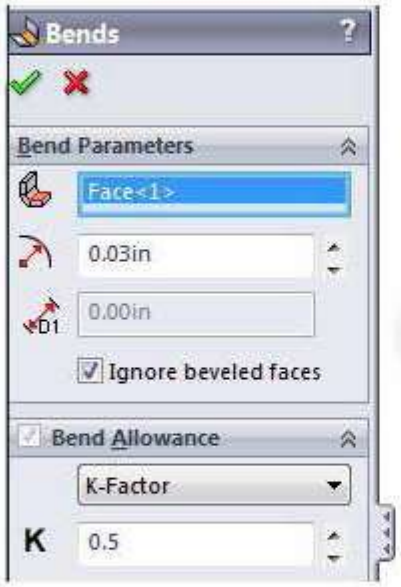






15. Click Sheetmetal>Insert Bends, 

click flat face as reference when it flatten. Set bend radius to 0.03in and K factor 0.5 and OK.



16. Your simple Sheetmetal bend is ready. Look at part tree.

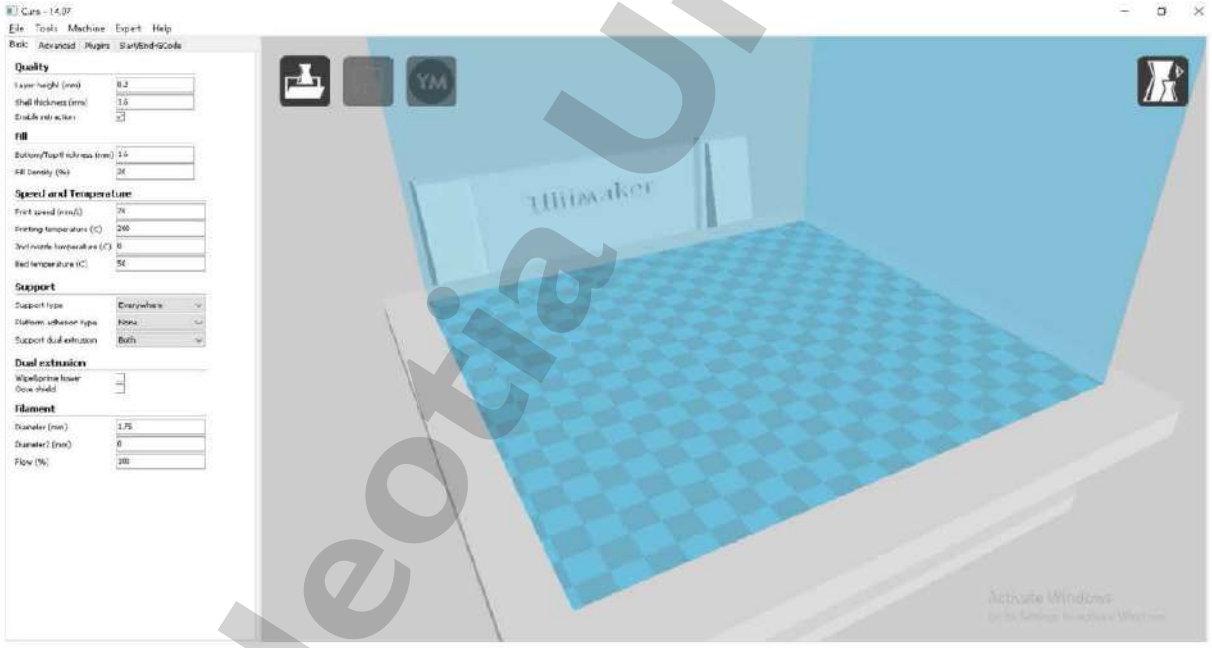
EXPERIMENT -5

To Create University Name using Ultimaker Cura

Step 1: first we download the logo of our university through internet

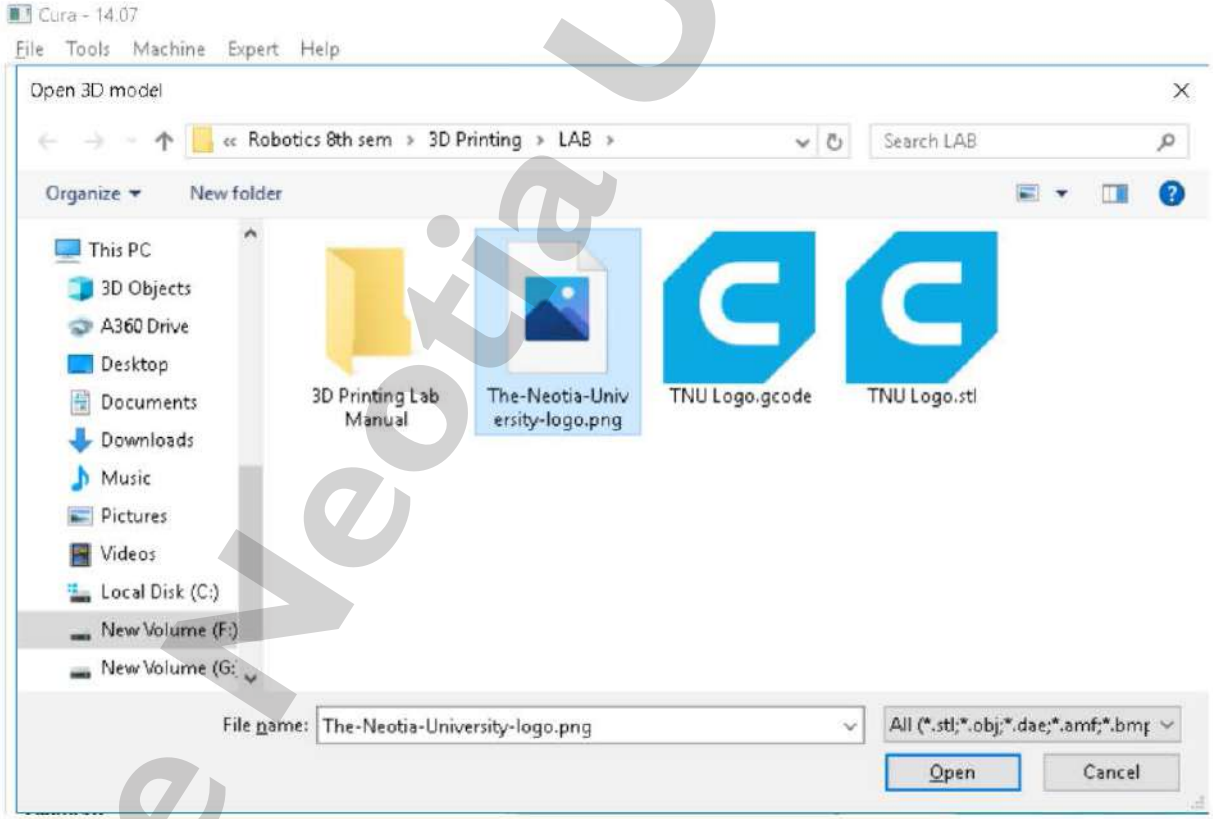
THE NEOTIA UNIVERSITY
ज्ञानम् आत्म प्रदीपाय

Step 2: here we used cura 14.07 as CAD software. There we open the software.

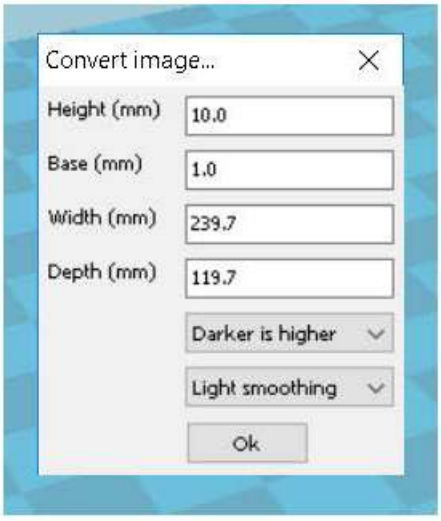


Step 3: Open file to load Model

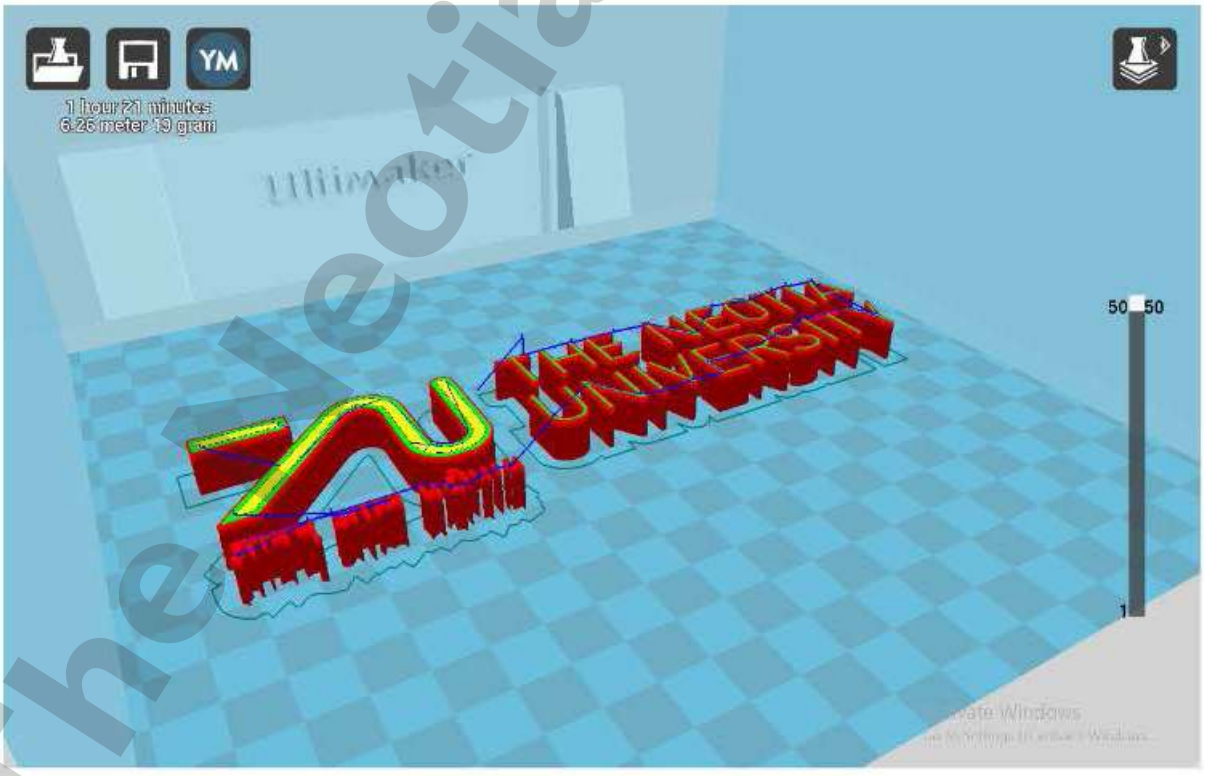
Step 4: we choose the logo from Load model file and open it



Step 5: Select specific parameters



Step 6: it will give an out put like :



Step 7: it will also generate g code like

```
;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_hei
;Print time: {print_time}
;Filament used: {filament_amount}m {filam
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment
;M109 S{print_temperature} ;Uncomment to
G21         ;metric values
G90         ;absolute positioning
M107        ;start with the fan off
G28 X0 Y0   ;move X/Y to min endstops
G28 Z0      ;move Z to min endstops
G1 Z15.0 F{travel_speed} ;move the platfo
G92 E0                   ;zero the extrude
G1 F200 E3               ;extrude 3mm of f
G92 E0                   ;zero the extrude
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...
```

Step 8: Get Ready to Print

Insert the card containing g code in your computer. Then drag the gcode file that you just save into the card.

Step 9: Let's Print!

Insert the small chip in the 3D printer. Select your file and print it. If you want you want to stabilize your logo, you can make a base for it by following the same procedures of making logo.