

FREECAD DESIGNING & 3D PRINTING

User Guide

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1.FREECAD

1.1 Designing:



- In FreeCAD software select Part Design Workbench, click on create new sketch and choose XY plane to sketch the 2D figure of the required 3D model.



Fig: Drawing Tools



Fig: Constraint Tools

- Choose line tool from drawing tools and draw the required 'T' shape 2D diagram.
- Now set the required constraints of the figure using constraint tools.

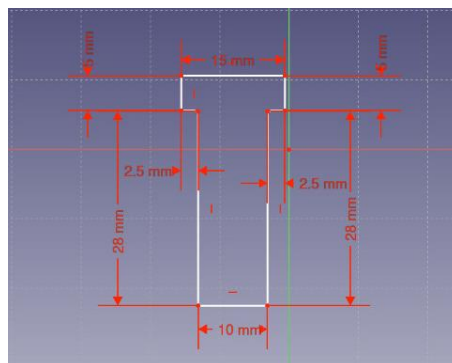


Fig: Sketch View

- Update the sketch and select the sketch and pad it for 3D Sketch.

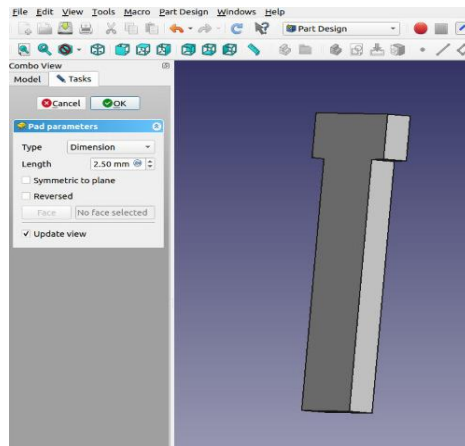
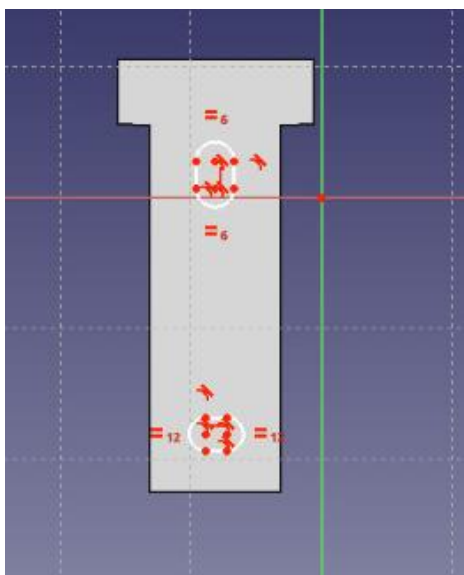


Fig: 3D View

- Select the top face of padded sketch and click on create sketch.
- On selected face draw the required sketch and do the pocketing.



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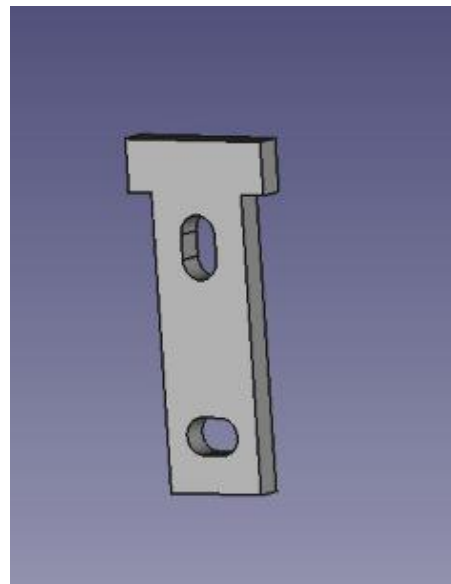


Fig: Pocketed sketch

- After updating it, select the body of sketch and from File menu Export it to ".stl" file and save it.

2. CURA

- Open Cura Software and open “.stl” file in it, then set infill to 50.
- Slice and generate the G-code and save it as “.gcode” file, which is used to print.

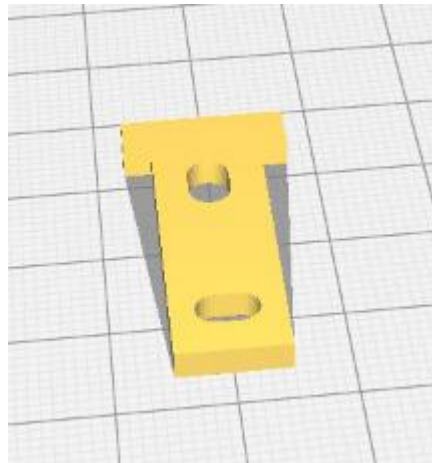


Fig: Preview

3.3D Printing

- Select the “.gcode” file in printer and click on print.



Fig: output print.