Julia Advanced 3D Printer - Usage Information Booklet

Wadhwani Electronics Lab

Department of Electrical Engineering

INDIAN INSTITUTE OF TECHNOLOGY BOMBAY

Rabiya Kunhayi	
Shantanu Singh	213079012
Sayali Duragkar	203070025
Goutham A P	193079004

Contents

1 Computer Aided 3D Designing

1.1 Introduction

The STL (Standard Triangle Language) is the industry standard file type for 3D Printing. It uses a series of triangles to represent the surfaces of a solid model. All modern CAD (Computer Aided Design) software allow you to export their native file format into STL. The 3D model is then converted into machine language (G-code) through a process called "slicing" and is ready to print. [**?**]

Few popular examples of 3D designing softwares are:

1. **Solidworks** :- `https://www.solidworks.com/`
2. **Fusion 360** :- `https://www.autodesk.com/products/fusion-360/`
3. **Solid Edge** :- `https://solidedge.siemens.com/en/`

1.2 Fusion 360

Fusion 360 is a cloud-based CAD/CAM/CAE tool for collaborative product development. Fusion 360 combines fast and easy organic modeling with precise solid modeling, to help you create manufacturable designs.[**?**]

Fusion 360 is free to use for students who register in the **Autodesk Education Community**:- `https://www.autodesk.com/education/edu-software/overview?sorting=featured&filters=individual`. Few Important Links are listed below.

- **System Requirements**:- https://knowledge.autodesk.com/support/fusion-360/learn-explore/caas/sfdcarticles/sfdcarticles/System-requirements-for-Autodesk-Fusion-360.html
- **Documentation**:- `https://help.autodesk.com/view/fusion360/ENU/`
- **Tutorials & Self-guided Courses**:- `https://help.autodesk.com/view/fusion360/ENU/courses/`

Few Important Concepts to begin with in the https://help.autodesk.com/view/fusion360/ENU/:

- Use the Sketch tools to create the sketch profiles that drive the shape of solid, surface, and T-Spline bodies in your design. Sketches are the underlying geometry that support the creation of 3D solid, surface, and T-Spline bodies in your design. Ex: Lines, Shapes, Splines, Mirrors and Patterns
- The Solid tab contains traditional solid modeling tools within the Design workspace in Fusion 360, and supports both parametric and solid modeling modes.

- The following commands in the Design workspace, in the Solid > Create panel, let you create a solid body from a closed sketch profile, open sketch curve, or planar face in Fusion 360.
  - Extrude
  - Revolve
  - Sweep
  - Loft
  - Shell
  - Fillet
  - Champer
  - Sweep

After designing, export the STL file and import it into the slicer software for 3D printing. There are many websites where we get STL files of popular designs, for ex:**Makerbot Thingverse**:- `https://www.thingiverse.com/`

2 Fracktory Slicer Software

A slicer is computer software used in the majority of 3D printing processes for the conversion of a 3D object model to specific instructions for the printer. In particular, the conversion from a model in STL format to printer commands in g-code format in fused filament fabrication and other similar processes. [**?**]

The slicer first divides the object as a stack of flat layers, followed by describing these layers as linear movements of the 3D printer extruder, fixation laser or equivalent. All these movements, together with some specific printer commands like the ones to control the extruder temperature or bed temperature, are finally written in the g-code file, that can afterwards be transferred to the printer.

1. Fracktory is a 3D slicer application developed by FRACKTAL WORKS supported by the 3D printer.

2. Fracktory software imports the 3D design in .stl/.obj file format (preferably .stl as it can be edited). Note that the Fracktory software is used to print the model not for making the model.

3. The model is viewed in 3D with bottom plane is build-plate of the printer.

4. In 3D printing parameters settings, both recommended and customised options are available. In recommended settings, the parameters are set automatically based on the design imported and on the other hand, we can set each parameter manually based on the understanding and experience in the 3D printing.

5. Various printing parameters like Infill, Building Support etc. can be set to preferred values and more information about each one of them can be obtained by hovering over **'i'** symbol at each setting type. For more information, refer to **10 Most Important 3D Printer Slicer Settings**:- `https://all3dp.com/2/3d-slicer-settings-3d-printer/`

6. After finalising all the print parameters, click on 'Prepare' option at the bottom-right corner. The estimated time to print the model is shown and the parameters can be optimised according to the necessity.

3 The Julia Advanced 3D Printer

An Introduction to Julia Advanced is given in the link:
Introduction Video:- `https://youtube/FDG05gGNbcc`.

About the printer:-

1. Manufactoring Company :- FRACKTAL WORKS
2. Model :- Julia Advanced
3. Nozzel Size :- 0.4 mm Nozzle
4. Layer Resolution :- 100 - 300 micron
5. Supported Materials :- PLA, ABS, EVA
6. Maximum Dimensions of Model:- 200mm * 200mm *200mm
7. Printing Speed:-
   (a) Normal:- 50-55 mm/s
   (b) For Fine Printing :- 45 mm/s
   (c) Fast :- 60-65 mm/s

Other specifications are mentioned in the link below.

The Brochure for the Julia Advanced Printer can be found in the following link: `https://5.imimg.com/data5/BK/ID/TH/SELLER-20503702/3d-printer-julia-advanced.pdf`

4 Steps to follow from STL to 3D printing

1. Select the 3D Printer

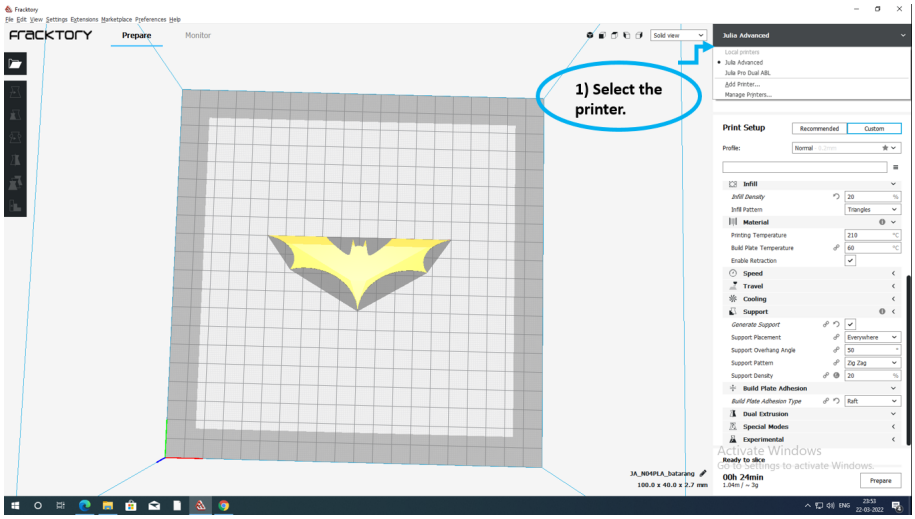

Figure 1: Select the printer.

2. Select the material used for printing.

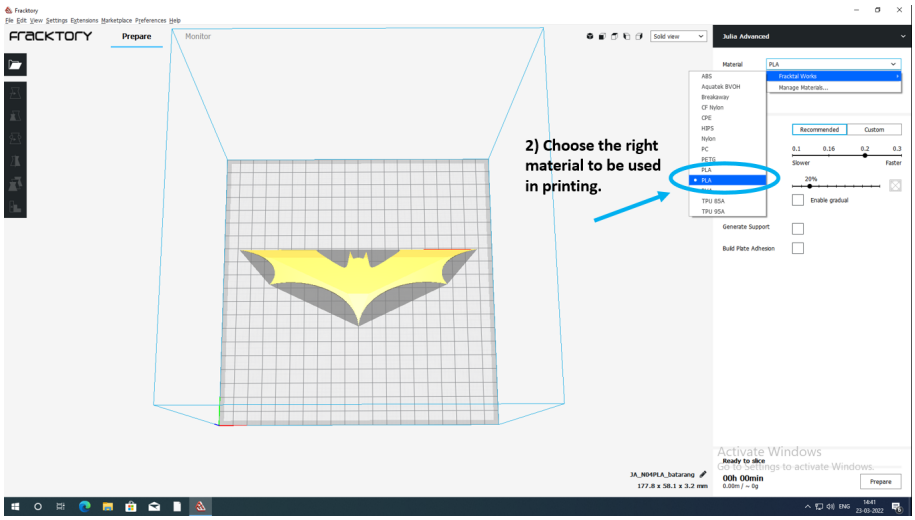

Figure 2: Select the material.

3. Check whether the selected nozzle size is same as of the printer.

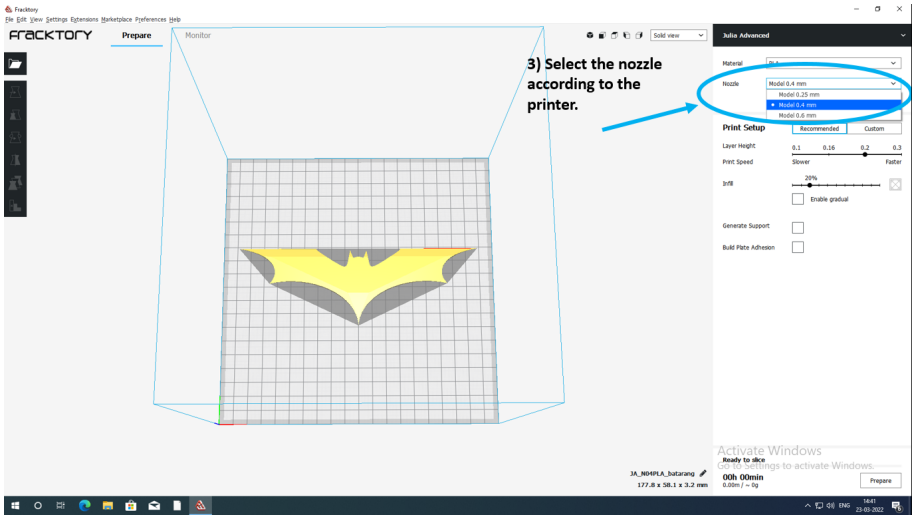

Figure 3: Select the nozzle size

4. Import the .stl file into the Fracktory software.

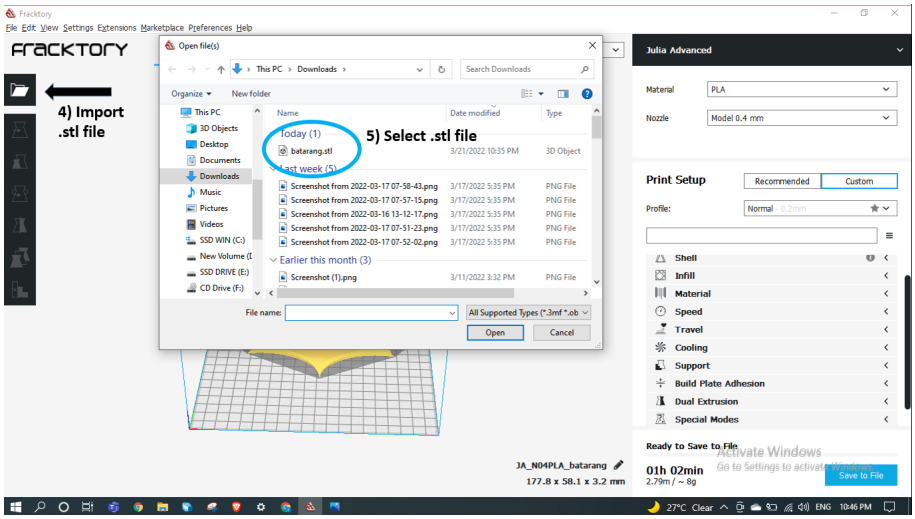

Figure 4: Opening the .stl file

5. Set the 3D printing parameters according to the requirement. Beginners are encouraged to use recommended settings. Tick the Build plate adhesion option in the right pane.

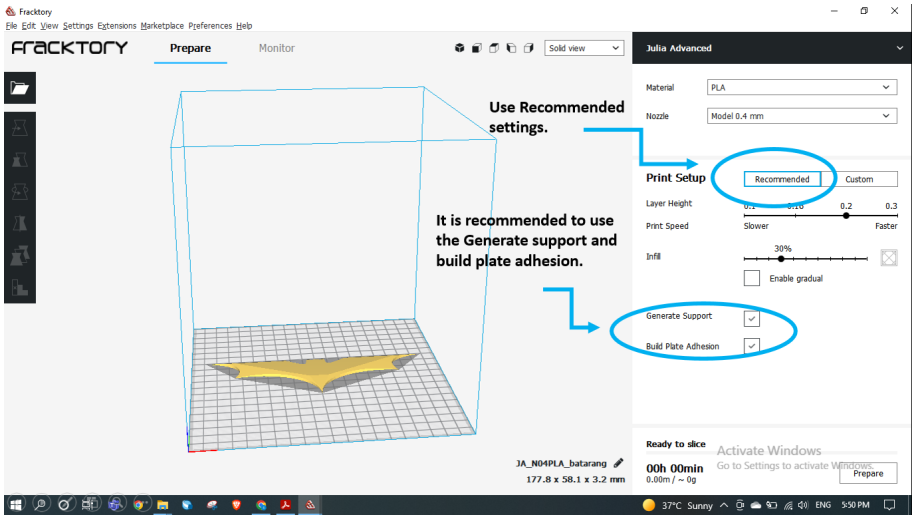

Figure 5: Select the setting(Recommended).

6. If one wants to do customize settings then they are encouraged to change the wall thickness according to requirement and can choose the infill pattern.

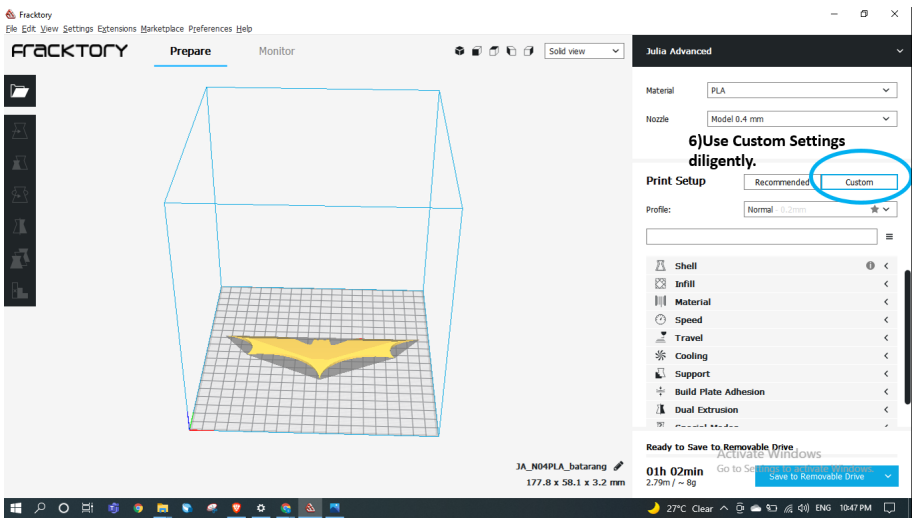

Figure 6: Selecting the setting(Custom).

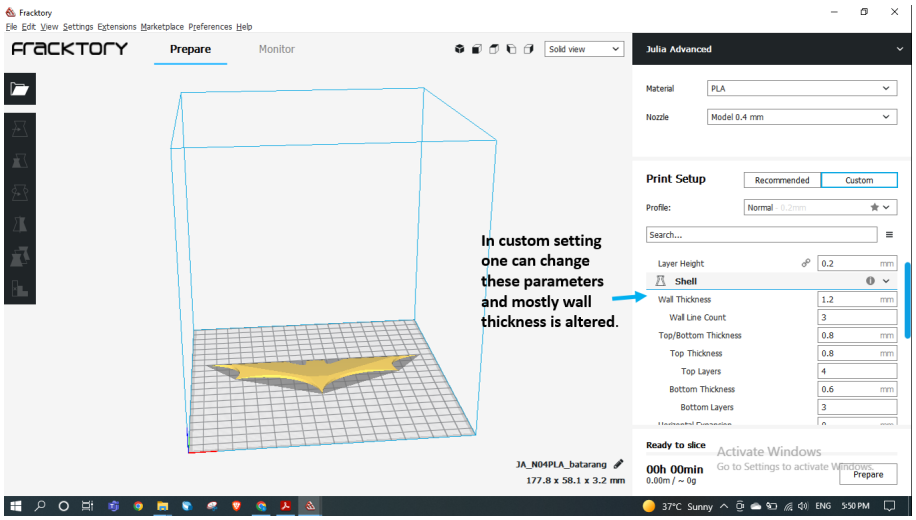

Figure 7: Wall thickness option.

7. The built-plate adhesion type among brim,raft,skirt can be chosen based on model otherwise it is mentioned in recommended settings.

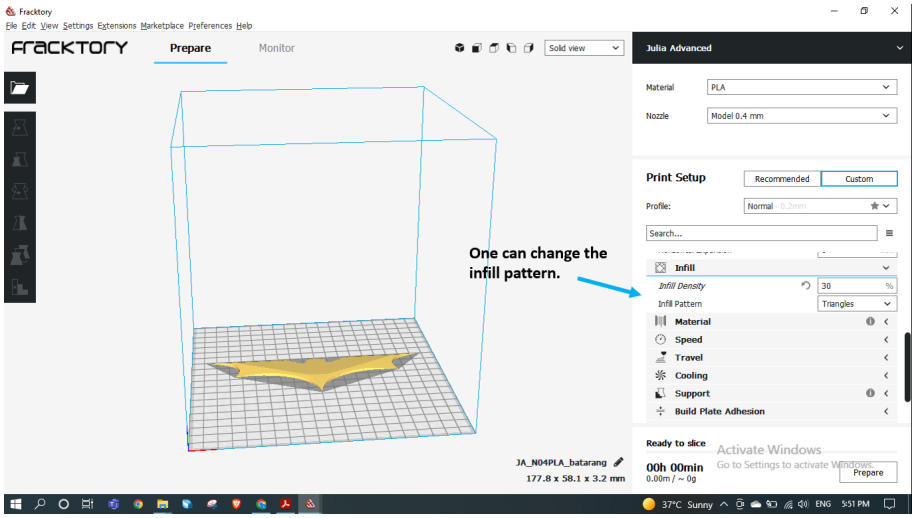

Figure 8: Option for Infill.

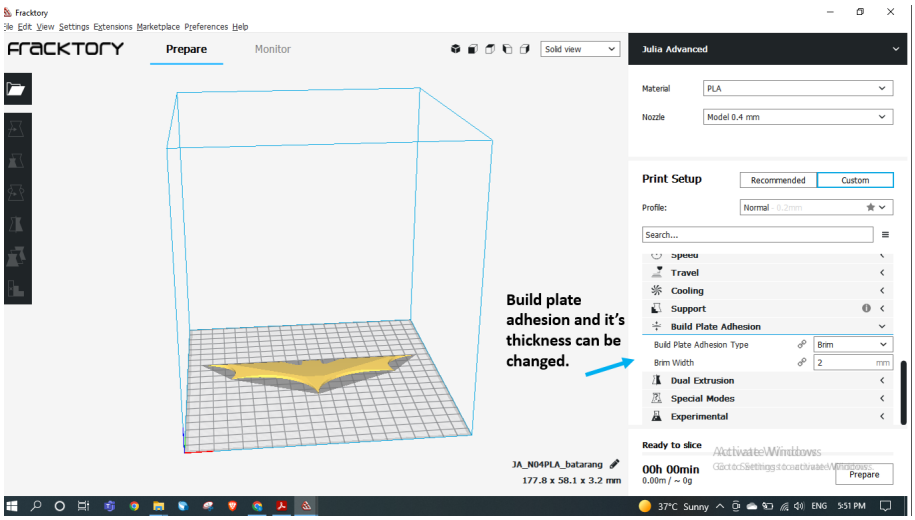

Figure 9: Build plate adhesion type and it's width.

8. After setting the print parameters, prepare .gcode file.

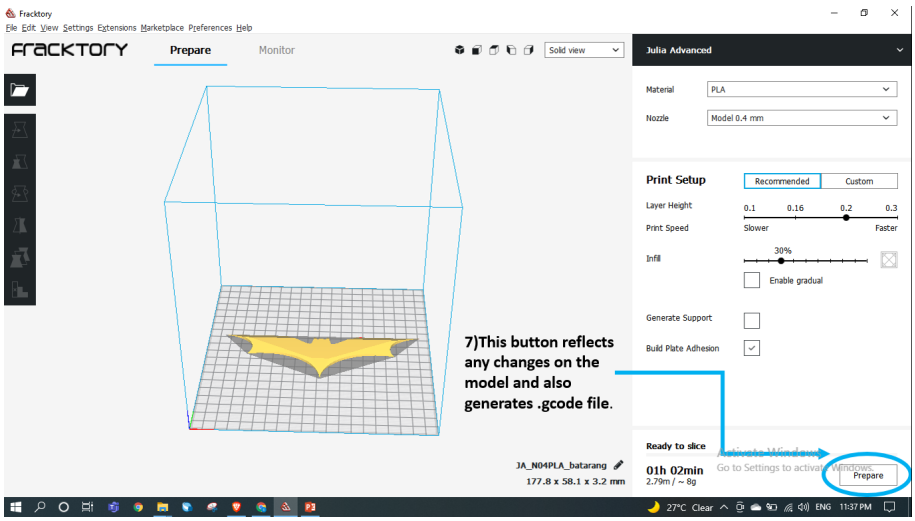

Figure 10: Prepare the model.

9. Export the generated .gcode file to a USB memory device.

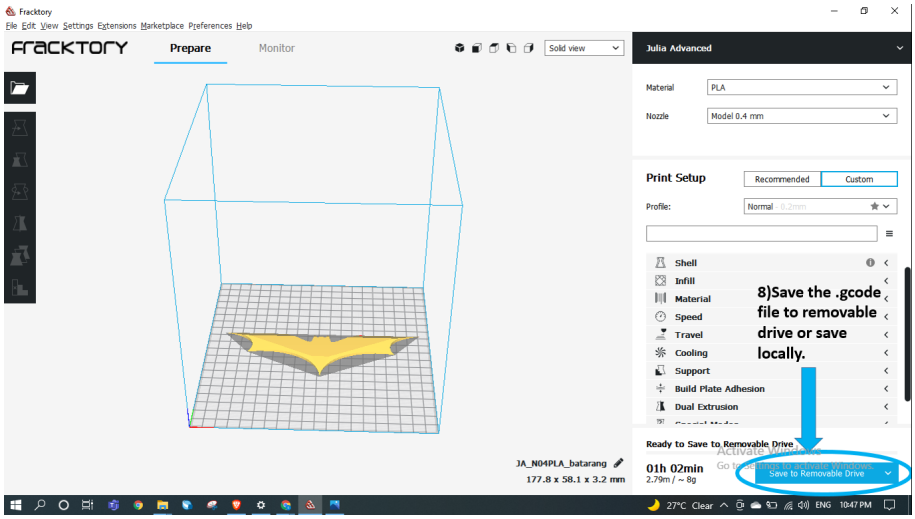

Figure 11: Saving the model in USB memory device

10. Mount the USB memory device to the 3D printer (Julia Advanced) and select the model to be printed.

Figure 12: Select the model to be printed on the USB device.

11. Choose the storage(in this case removable disk) where the g-code file is stored.

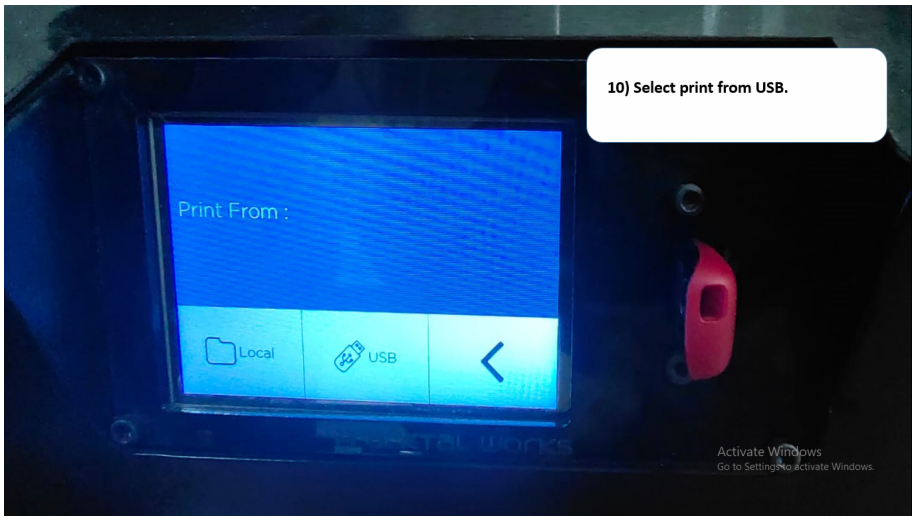

Figure 13: Select the USB option.

12. Select the model.

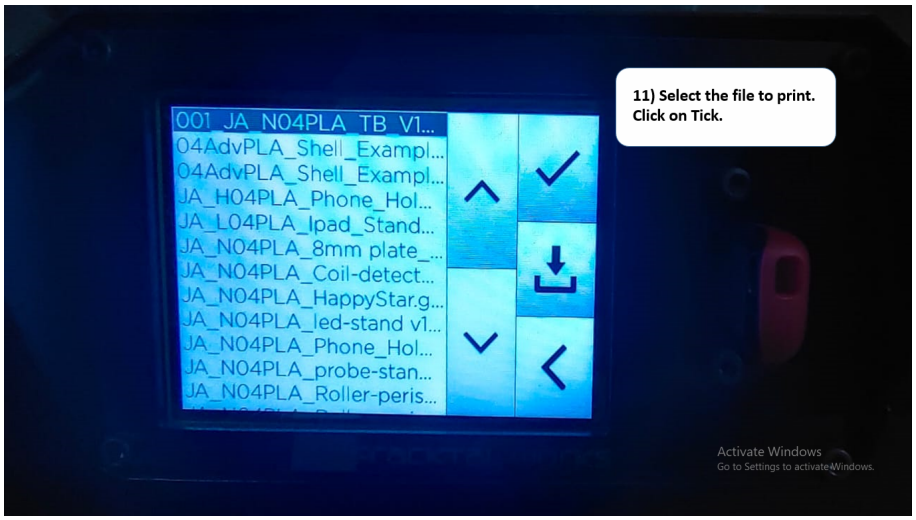

Figure 14: Review the model.

13. Print.

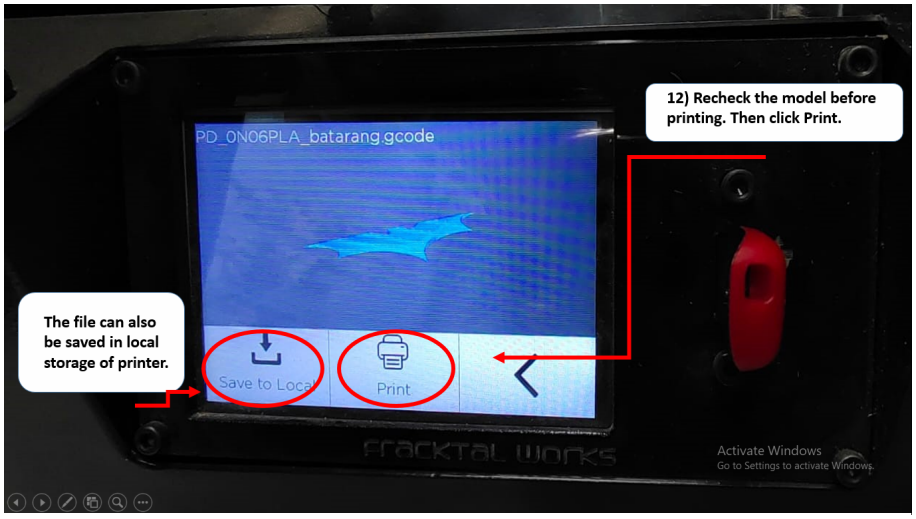

Figure 15: Printing starts.

14. Before printing the model the temperature of nozzle and build-plate rises to a required level.

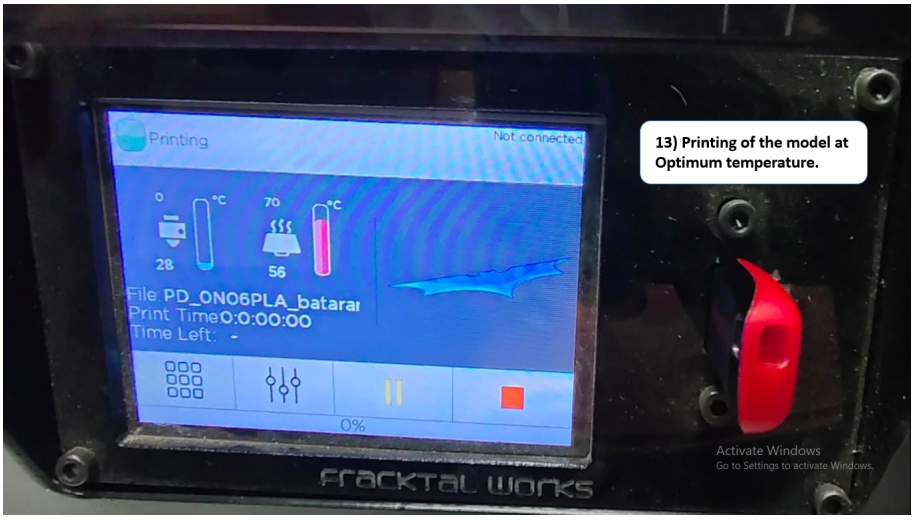

Figure 16: Temperature of the nozzle(left) and build-plate(right).

15. After giving the print command the printer does the 9-point calibration by moving the nozzle at different positions on the built plate.

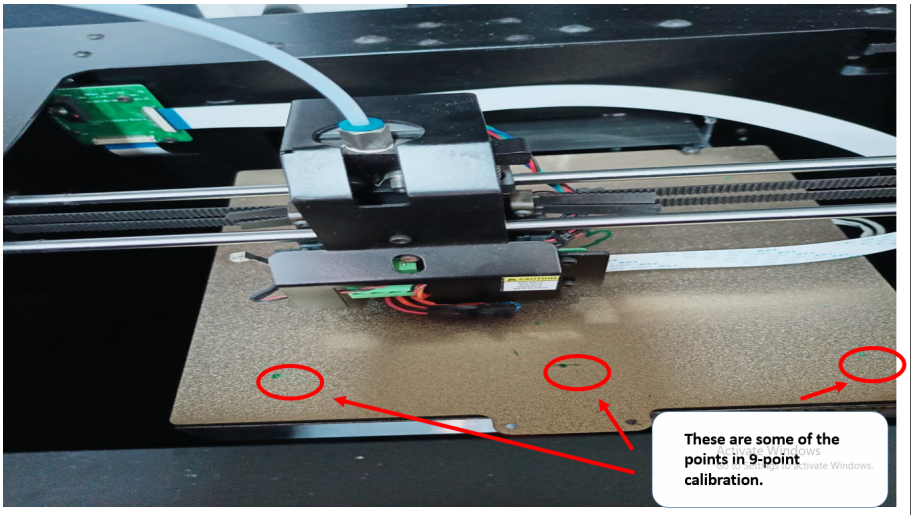

Figure 17: 9-point calibration of printer.

16. Take the model out of the printer only after the minimum temperature of built plate and nozzle is achieved.

Figure 18: Model just after removing from printer.

17. Remove the brim of the model with the cutter.

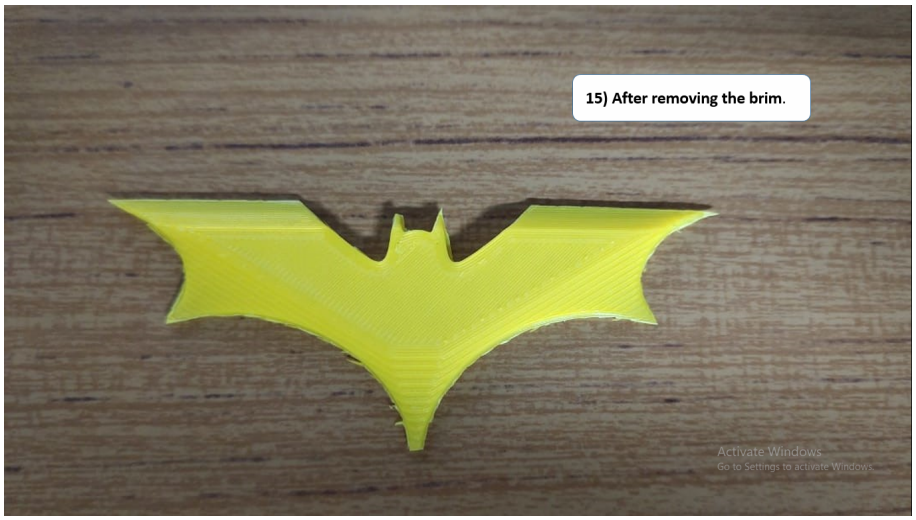

Figure 19: Printed model after cutting the brim.

5 Calibration

Calibration is one of the important step in the process of 3D printing. If the printer is correctly calibrated then one can generate the expected output minimising errors. Calibration involves built-plate leveling and nozzle cleaning. The following steps should be followed for calibration:-

1. On the printer menu go on calibration option.

Figure 20: Select the calibrate option.

2. Select the wizard menu.

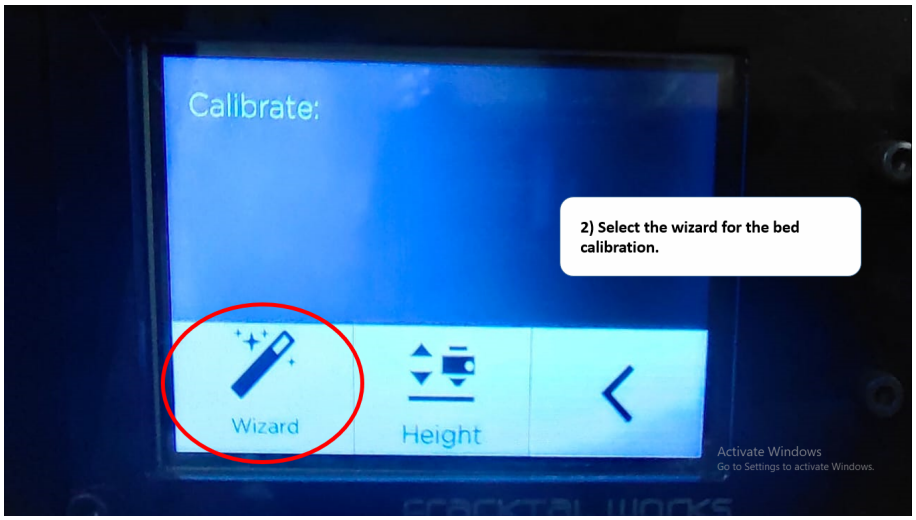

Figure 21: Select the wizard option.

3. Select 'Quick Calibration' and the built-plate will rise and nozzle will move through 3 points left, right and back.

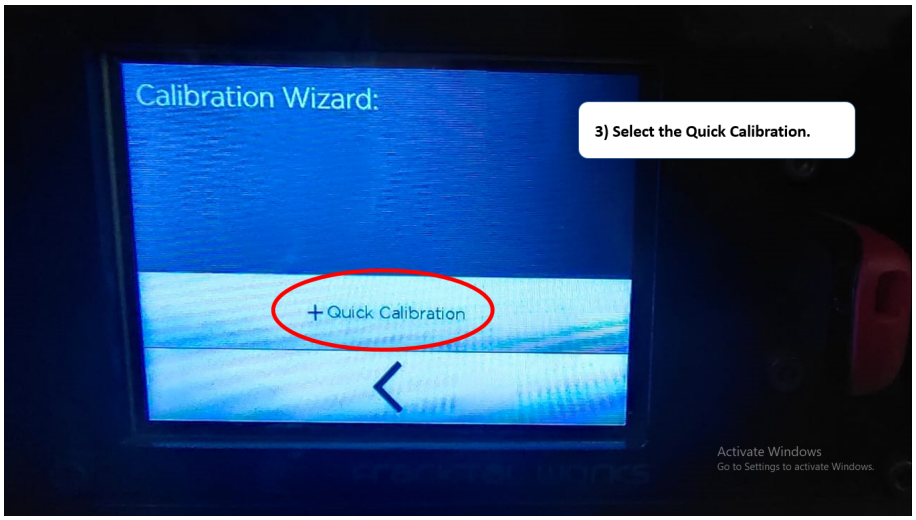

Figure 22: Select the quick calibration.

4. As the nozzle moves at the first point, tighten or loosen screw at the bottom of built-plate exactly below the position of the nozzle until the Red LED light at bottom of built-plate stops blinking and glows constantly.It may happen that Red LED may not be glowing so try to adjust the screw in both direction so that it starts blinking and then glows solidly.

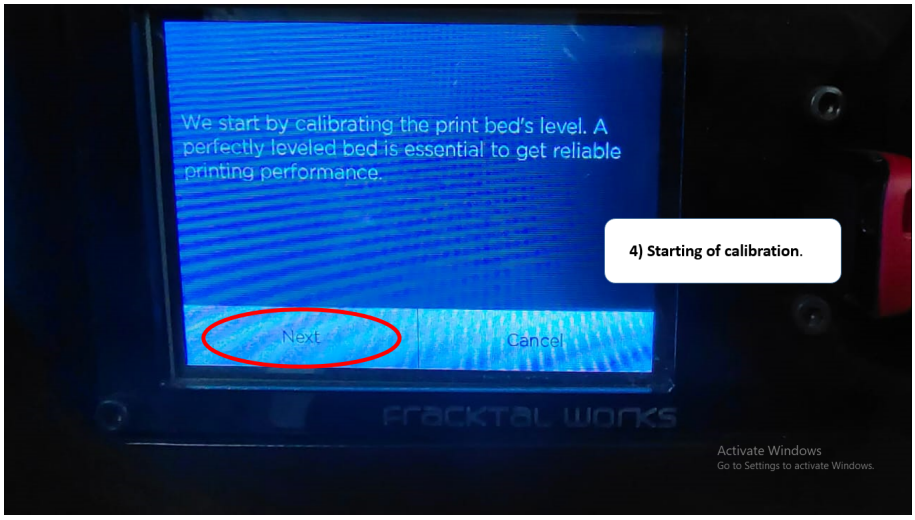

Figure 23: Calibration begun.

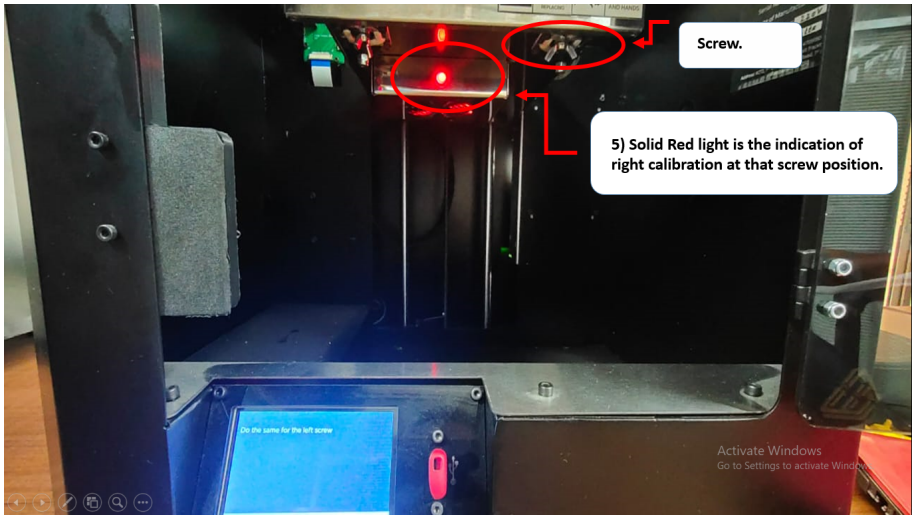

Figure 24: Adjust screw up and down red led goes from blinking to solid red.

5. Then click on 'Next' on the screen and repeat Step 4 for the next point. Repeat the same until all the three points are finished and then click on 'Done' on the screen.

6. For nozzle cleaning, take a few drops of acetone on a cotton ball and remove the filament at nozzle mouth. **Warning: The Nozzle temperature should be low while doing this, which can also be seen at the screen.**

7. For removing extra filament from nozzle mouth, go to Control Menu → Load Filament option and remove filament until it comes out smoothly for better printing.

8. Calibration should be done once in a week and **make sure that the temperature of both nozzle and build-plate are low**

9. 3D printer should be switched off once the model is removed after complete printing.

6 Precautions for operating the Julia Advance 3D printer

1. Check whether the material (e.g PLA) used in the the printer is same as specified in the Fracktory software.

2. The calibration of the nozzle and the built plate is important otherwise printer will not design the shape correctly.

3. The nozzle and built plate must be clean, dust particles may deteriorate the model.

4. The nozzle and built plate are to be preheated to certain temperature to make the material flow smooth.

5. Initially, run the nozzle for some time to remove some of the material ensuring smooth flow for the design.

6. Before printing, ensure if the model to be printed and the model in the display of the printer monitor are same. Sometimes, it may take hours to print the model.

7. The time for printing and overall time is mentioned on the printer display when model is in process of printing , model should be removed only after the overall time is passed.