|| Volume 5 || Issue 12 || April 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH & ENGINEERING TRENDS
Multidisciplinary Journal
Double-Blind Peer Reviewed Refereed Open Access International Journal

Exploring 3D Printing using CURA: A Slicing software

Gourav Dhore[1], Rushikesh Jagtap[2], Sumit Bhakad[3], Pratik Yadav[4], Mr. A.B. Sutar[5], Ms. Sudeshana Sawrate[6]

Student, *Department of Mechanical Engineering, AISSMS Polytechnic, Pune, India[1 2 3 4]*

Lecturer, *Department of Mechanical Engineering, AISSMS Polytechnic, Pune, India[5, 6]*

-- ***---

Abstract: Fused Deposition Modeling (FDM) is an Additive Manufacturing technique for printing 3D objects layer by layer. For forming layered structure initially, we need to form slicing path using CAM tool. With slicing tools, it is possible to convert digital 3D models into printing instructions for 3D printers. The general approach is: The model is cut into horizontal slices which are then used to create extrusion paths similar to milling paths in the traditional CNC field, which are then being filled with material, mostly plastic material. The goal of this study is to compare available slicing tools for 3D printers The main contributions of this study are: collecting methods and tools to judge print results, and exploring the CURA slicing tool through its applicability.

Keyword: *Cura, Repramp, CAM, G-code.*

-- ***---

I INTRODUCTION

1.1 3D Printer

A 3d printer is an additive manufacturing technique where 3D objects and parts are made by the addition of multiple layers of material. It can also be called as rapid prototyping. It is a mechanized method where 3D objects are quickly made as per the required size machine connected to a computer containing blueprints of any object. The additive method may differ with the subtractive process, where the material is removed from a block by sculpting or drilling. The main reason to use 3d printer is for 90% of material utilization, increase product life, lighter and stronger. 3D printing is efficiently utilized in various fields such as aerospace, automobile, medical, construction and in manufacturing of many household products.

1.2 Motivation for the present research work

Since over a century the visual world of printed scriptures has been dominated by the 2-D printing methods. Be that easy to read or comprehend but when it comes to imaging of definite and real-life models it is sorely outsourced. Any 3-D model cannot be represented and displayed easily in a 2-D workplace. The only thing worth mentioning for likable perception is the rendering of the image. This ushered in the era of the much-needed idea of "3-D" printing. Basically, the singular purpose for the division of 3-D printer was to prepare 3-D samples directly on the bed of the printer. It has been an effective way of manufacturing since many companies are now opting for this type of method for their production operations.

1.3 Process of 3d printing:

3D printing process can be described and defined in the following steps:

i. CAD Model Creation:

Initially, the item to be 3D printed is designed utilizing Computer-Aided Design (CAD) software. Solid modelers, for example, CATIA, and SOLID WORKS have a tendency to represent 3-D objects more precisely than wire-frame modelers, for example, AutoCAD. This procedure is comparative for the majority of the Rapid Prototyping building methods.

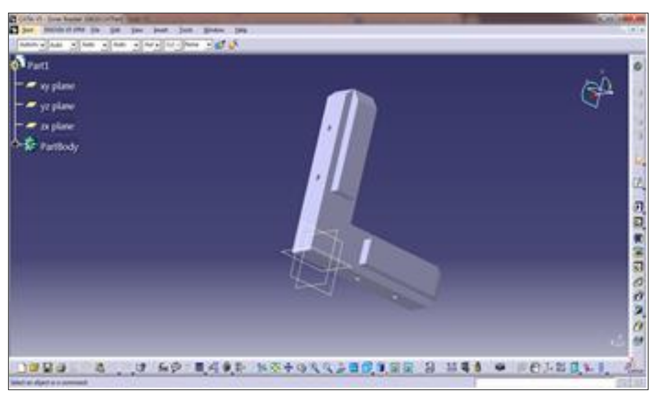


Fig 1 CAD model of L-shape Inner bolt

ii. Conversion to STL Format: The different CAD models use different methods to present solid parts. To have consistency, the stereo lithography format has been followed as the standard of the 3D printing industry.

|| Volume 5 || Issue 12 || April 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH & ENGINEERING TRENDS
Multidisciplinary Journal
Double-Blind Peer Reviewed Refereed Open Access International Journal

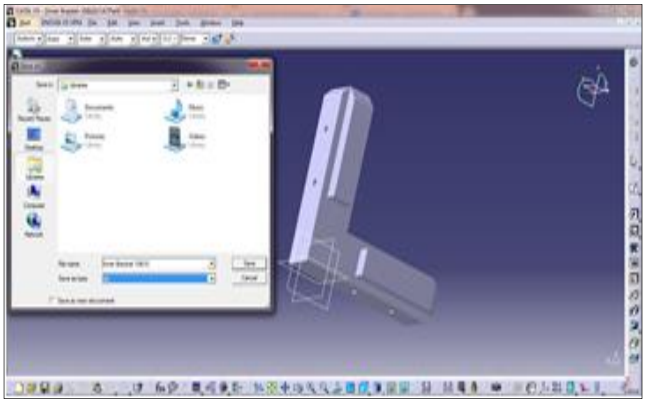


Fig 2 Conversion to STL Format

iii. Slice the STL File: A preprocessing computer program is done which readies the STL format going to be built. Numerous programs are there, which permit the user to tweak the model. The preprocessing program cuts the Stereo lithography model into numerous layers from 0.01 mm to 0.7 mm thickness, in view of the building method. The program likewise makes an auxiliary structure to help the model amidst of building. Sophisticated structures are bound to use auxiliary support.

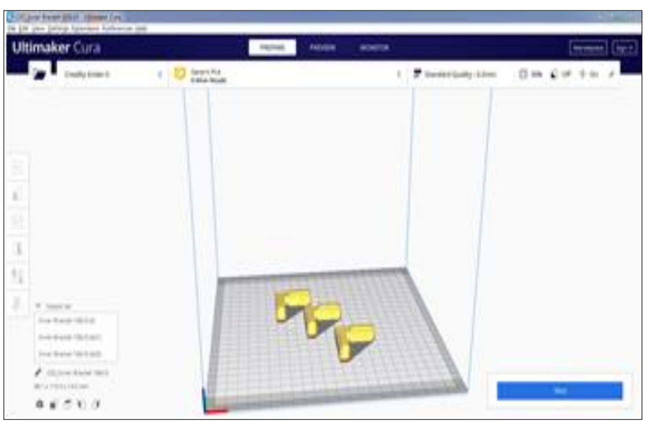


Fig 3 Slicing of STL File in CURA

iv. Layer by Layer Construction: The fourth step is the actual construction of the part. Using one of various techniques RP machines build one layer at a time from polymers, or powdered metal.

Fig 4 Layer by Layer constructed parts

v. Finishing: The printer produced resolution is very much sufficient for many of the applications but the printing will be a slightly oversized version of these desired object which can be the standard resolution and then the process of removing material can give greater precision. Some printable polymers allow the surface finish to be smoother and improved using chemical vapor processes.

1.4 Future scope

In future 3D printers would be accessible at a very low cost and can be even used in household applications as it would be reasonable and also very accurate and a multicolor extruder can be used and also the printers would be made movable. In India 3D printing is an exciting and interesting aspect as it is completely new concept.3D printing in rural background is being developing out of India. Most of leading industries are looking forward to have effective printers in their system. Currently 3D printing is at its peak of developing phase. The upcoming years will see a lot of happening in this market that would give a perfect guidance to the organization for the further advancement.

II LITERATURE SURVEY

2.1 Various techniques developed for additive manufacturing

2.1.1 Photo polymerization

Curing of photo reactive polymers/ resins with laser or UV light (Ex SLA, MATERIAL JETTING, TPP) [1] **Stereolithography Apparatus (SLA)** method was invented in 1986 and was typically used in the first-generation commercial 3D printers. Printers which are using stereolithography to concentrate the beam of UV rays on the top of the surface of the object which should be replicated. The object is filled with resin. When light hits the resin, you get a high resolution 3 D model of the object you have used. [2]

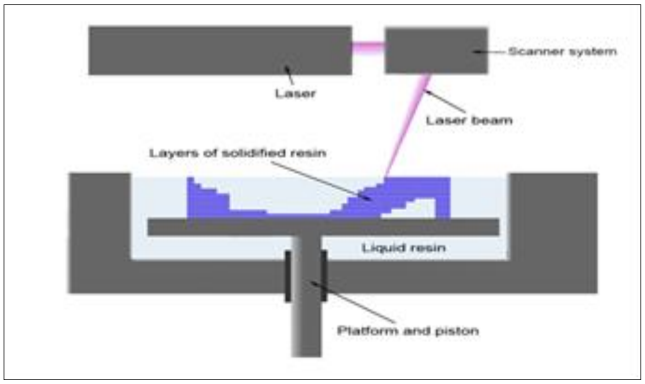


Fig 4 Stereolithography Apparatus (SLA)

The Material jetting is one of a unique and the only additive manufacturing technology that can be a combination of many different print materials within the same 3D printed model inside the same print job. The multi material is obviously a printing process is very much capable of constructing functional assemblies which reduces the need for multiple builds. With the respective ASTM standard of material jetting is the only process in which where there are some of the droplets that can build material can be selectively deposited onto a heated bed to develop a 3D object.

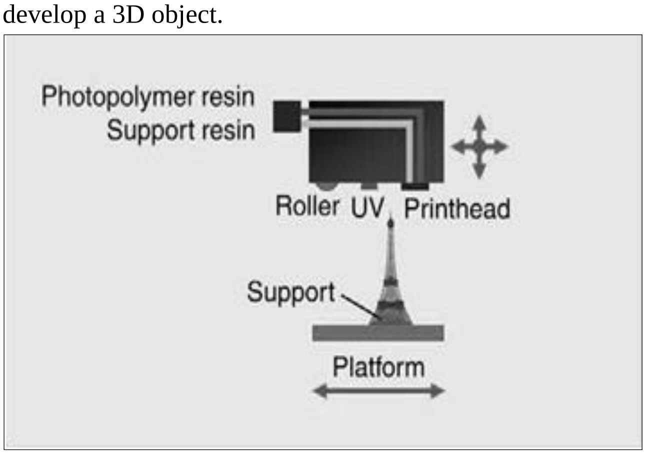


Fig 5 Material Jetting

Two photon Polymerization (TPP) is a promising three-dimensional micro fabrication method that has recently attracted considerable attention is based on two photon polymerizations with ultra-short laser pulses [2]. It is determined that when it is focused into the volume of a photosensitive material the pulses initiate two photon polymerization two photon absorption and subsequent polymerization [3].

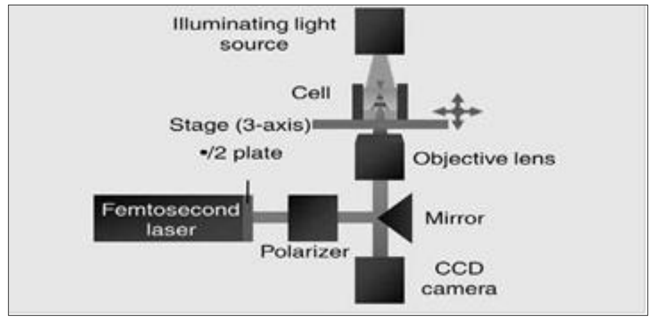


Fig 6 Two photon Polymerization

2.1.2 Powder:

High power laser to sinter small particles of material (Ex: SLS, BINDER JETTING)

Selective Laser Sintering (SLS) is a rapid prototyping process that builds media in powder form, which is fused together by using powerful carbon-dioxide laser to form final product. SLS uses a high-powered C02 laser to fuse small particles of powdered material to create 3 dimensional parts. When a laser where it will selectively which will fuse the powdered materials by scanning X&Y cross sections on the top of the surface of a powder bed. The model is built one layer at a time from supplied 3D CAD data [4]. SLS is very much capable of producing very highly durable parts for real world testing [12].

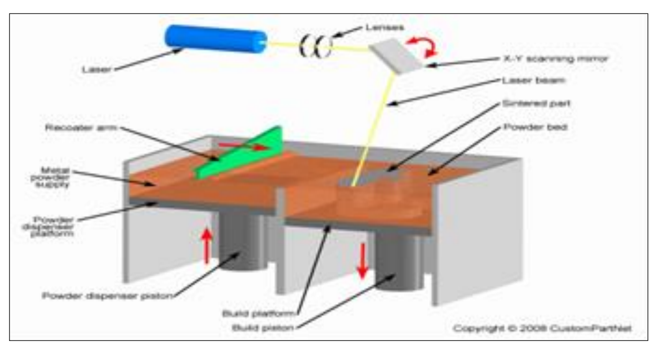


Fig 7 Selective Laser Sintering

Binder Jetting is a rapid prototyping where the material being jetted is a binder, and is selectively sprayed into a

|| Volume 5 || Issue 12 || April 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH & ENGINEERING TRENDS
Multidisciplinary Journal
Double-Blind Peer Reviewed Refereed Open Access International Journal

powder bed of the part material to fuse it a layer at a time to print the required part.

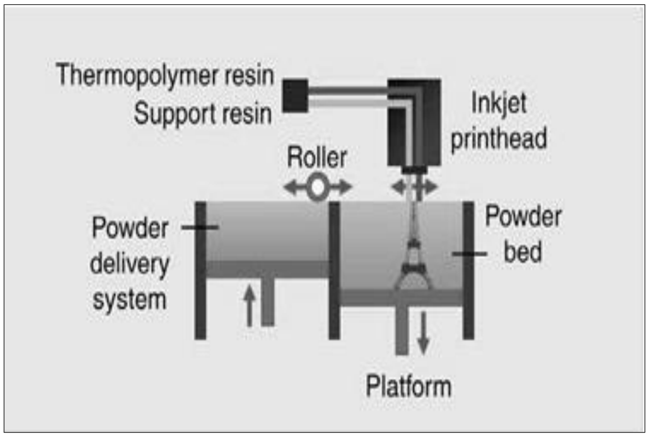


Fig 8 Binder Jetting

2.1.3 Material Extrusion: Extruding a filament at appropriate temperature. (Ex: FDM)

Fused Deposition Modeling (FDM): This is a process by which a machine deposits a filament (Thermoplastics or wax) [11]. On top or next to same material, in order to create a joint by heat or adhesion [4].

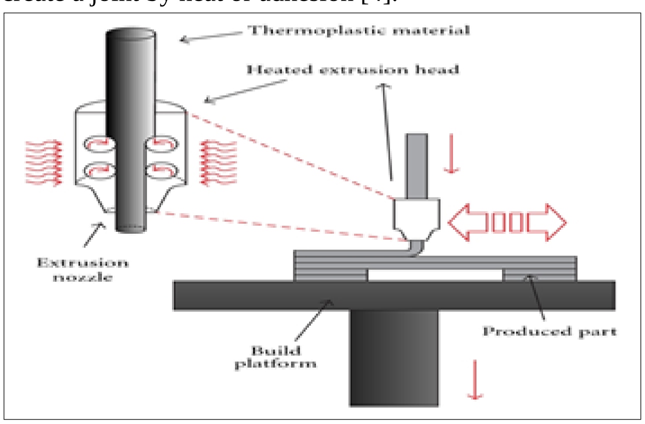


Fig 9 Fused Deposition Modelling

III. EXPLORING CURA

3.1 The normal workflow of 3D printing is:

Creating the three-dimensional model to print and exporting it and calculating the printer tool paths based on the three-dimensional model and exporting it (e.g., to G-Code) Printing, based on the control file. The process of calculating the printer tool paths based on the three-dimensional model file is called slicing. This is done by separate tools, so called slicing tools. To calculate these printer tool paths the slicing tools tries to find all solid shapes of the model. Therefore, the model has to be manifold, otherwise the slicing tool can- not find the solid bodies. Good slicing tools can try to repair the model if it is not manifold. The solid shapes are then cut into layers, as thick as the layer height of the printer. Now on every layer the necessary two-dimensional tool path to print this layer are calculated. This includes the movement on two axes and the feed rate of the extruder. Later the printer will print layer after layer two-dimensionally and move in the third dimension between the layers by the given layer height. There are various slicing tools with different advantages and disadvantages.

The quality of the slicing can have a huge impact on the printing result. A good slicing tool should not only calculate each layer separately but also look into the layers above and below. When dealing with overhangs and complicated structures the slicing tools can improve the print result a lot this way. As we could see in the produced G-Code the printhead was accelerated in areas where an overhang was detected and while bridging. Another reason indicating this behavior was seen as the start of a new layer was not put into the air but on existing material. Indication for looking above was found when analyzing the G-Code structure for infill patterns where the slicer looked a few layers ahead to adapt the infill pattern (closing the pattern for more stability). This case study's main focus is to show the difference between slicing tools and how they affect the printing quality.

3.2 Slicing tools

slicing tools calculate the printer tool paths from given three-dimensional mesh-based models. Input of the slicing tools is always a STL file, the output is always a G-Code file. In this section we present the slicing tools we selected for the tests and the selection criteria. Besides the slicing tools abilities their configuration also has a huge impact on the quality. In the section slicing tool configuration we explain the settings we use for our tests. For better comparability we use the same settings for all tests and slicing tools. All settings are optimized for our printer, a RepRap- Mendel printer. Our print results are depending on this printer and can be different on other printers.

|| Volume 5 || Issue 12 || April 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH & ENGINEERING TRENDS
Multidisciplinary Journal
Double-Blind Peer Reviewed Refereed Open Access International Journal

3.2.1 Available slicing-tools

Under the assumption, that the number of hits fora slic- ing tool in popular search engines is associated with the popularity, the following packages belong to the most popular slicing tools (as of May 2014):

1. Cura, (open-source project by Ultimaker)
2. KISSlicer, (commercial project)
3. Skeinforge, Release 50 (open-source project)
4. Slic3r, (open-source project)
5. RepSnapper, (open-source project)
6. Miracle-Grue/Makerware, (freeware project by Makerbot)

3.3 Selection criteria

From the available slicing tools found we use the following criteria for selecting the slicing tools to investigate in this study:

1. **Reliability:** The slicing tool must be able to handle all our test models.
2. **G-Code compatibility:** The G-Code must be compati- ble with the RepRap firmware so that it can be printed with the RepRap Mendel printer.
3. **Configurable:** To get comparable results with different slicing tools all of them must offer specific configuration properties. The properties “print temperature”, “print bed temperature”, “layer thickness”, “fill den- sity”, “print speed” and “minimum layer print time” must be configurable. In every test we configure all slicing tools with the same configuration values.

3.3.1 Excluded slicing-tools

1. **Miracle-Grue/Makerware**
   The G-Code generated with Miracle-Grue was not compatible with our RepRap-Mendel printer. Therefore, we had to exclude it was not possible to compare the results with the other slicing tools on our RepRap printer.
2. **RepSnapper**
   RepSnapper was excluded because it did not work reliably. When slicing the model DragonsEgg the slicer crashed repeatedly. When slicing the model TextTest not all parts of the model were sliced. RepSnapper is in an early development beta state.

3.3.2 Selected slicing tools

Within the scope of this work 4 slicing-tools are analyzed and described: Cura, KISSlicer, Skeinforge and Slic3r. The other tools are excluded due to their lack to fulfill the selection criteria.

1. Cura

Cura the open-source software developed by Ultimaker includes everything to prepare a 3D file for printing and slicing it. It is available on Linux, Mac and Windows. Multiple industry-standard files like STL, OBJ (Wave- front 3D file), DAE (Digital Asset Exchange) and AMF (Additive Manufacturing File) can be used. There are 4 simple standa profiles included. Cura has a user-friendly graphical interface; the buttons with main functionalities are well-arranged and mostly labeled.

2. KISSlicer

KISSlicer is a closed source slicing tool, there is a free version and which has all the features needed for a single-head machine which can be extended to PRO version with support of multi-head machines and multi-model printing. It generates G-Code from STL files. KISSlicer is available on FreeBSD, Linux, Mac and Windows. One can use KISSlicer in different languages. KISSlicer has a gray theme with orange-colored buttons.

3. Skeinforge

Skeinforge is a free open-source program. It is composed of Python scripts which generates G-Code instructions of a 3D model for RepRap. Skeinforge supports the file for- mats STL, GTS (GNU Triangulated Surfaces), OBJ, SVG (Scalable Vector Graphics), XML (Extensible Markup Language), G-Code and BFB (G-Code in the Bits from Bytes format). Skeinforge is more complicated to install and the user interface is less intuitive compared to other slicing tools. The better way to install Skeinforge is installing it with other host software which includes Skeinforge.

4. Slic3r

Slic3r converts STL, OBJ and AMF 3D models into G-Code instructions. It is available on Linux, Mac and Windows. Addition- ally Slic3r can be used from the command line. The GUI version provides a G-Code and model visualization as well as profiles and a configuration wizard. Slic3r is also integrated in various printer host software.

3.4 Support structures

In all tests we print without support structures. Our main goal is to look how good the slicing tools can handle difficulties without support. In general support structures decrease the print quality (arrears, imprints) so the results are best if a slicing tool succeeds printing a model without support structures. All test models are printed without raft (layers below the model to improve printbed adhesion). With good slicing tool configurations and a well-adjusted printer there is no need for a raft which also decreases the print quality.

3.5 Advanced settings

Most of the other setting values are set to the slicing tools default values. This might affect the printing results. In some cases, it would be possible to set settings to the same values but we keep the default values in order to harness possible internal slicer optimization. Slicer software is expected to have well-adjusted default settings in order to enable novice users printing desired results. We enable retraction (pulling back the filament when traveling) for all slicing tools.

3.6 Usability

User interface and configuration

This section analyses the user experience of the slicing tools– the user interface and the performance. The different slicing tools are executable applications – executable as command line application and/or as standalone application.

3.6.1 Usability of cura

Cura has a well-arranged 2-column layout. On the left side are the setting possibilities which are distributed in different tabs. On the right side is the view of the model and the buttons with main functionalities like loading and saving the model. On the view you can also edit the model with the following operations: rotate, scale and mirror. The switching of the view mode is also possible. There are for example the view mode Layers, Transparent, Overhang and Normal. If you select the Layers view mode you can see the movement lines of the extruder for each layer of the model. Cura offers a medium number of settings. There are less settings available compared to the other slicing tools. Its focus is on a user-friendly interface. Cura does not provide separate profiles for different categories but pro- file loading and saving for all settings as well as loading settings from Cura G-Code files.

3.6.2 Usability of KISSlicer

KISSlicer has partially a dark and gray theme. The main functions are orange colored so you can easily see them. KISSlicer provides 3 different levels of settings which are beginner, medium and expert. If you select one of these levels or change the current level the user interface refreshes itself directly. So the setting possibilities are limited to the selected level. The Reset-Button resets the view to the initial view. KISSlicer provides three different view modes – 3D model view, 2D G-Code layer view and a combined 3D model and G-Code view. In the advanced mode KISSlicer offers many detailed settings. It also provides separate profile switching for style, support, material and printer. This is very use- ful, for example for printing different materials with the same printer. Some default values are configured poorly for example the extruder speed is too high by default and we had problems with the pre-configured print bed roughness.

3.6.3 Usability of Skeinforge

Skeinforge has only a graphical user interface for the settings. It provides a command line interface which can be used by other tools for example Repetier-Host a host software for 3D printer. Skeinforge does not provide separate profiles for different categories but profile switching for all settings.

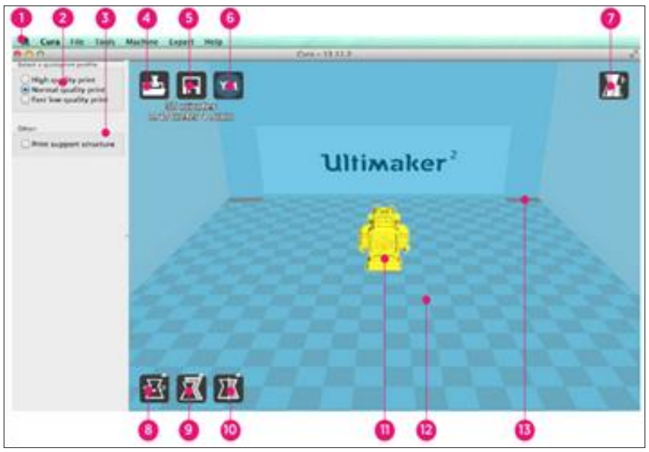
The strength of Skeinforge is the huge amount of set- tings which enables advanced users to adapt the slicing process very detailed.

3.6.4 Usability of Slic3r

Slic3r has a simple graphical user interface as well as a command line interface. The view of the model is very limited and the settings are kept simple. The interface is distributed in 4 tabs: Plater, Print Settings, Filament Settings and Printer Settings. Editing the model is difficult to get used to because of the complicated adjustment of the buttons. Slic3r provides only a 2D view of the edited or loaded model. There is a button which opens a window to show the model in 3D. It provides separate profiles for print, material and the printer. In the advanced mode Slic3r offers many detailed settings. There is also a configuration wizard for creating profiles for the most common printers.

CURA Software interface:

This is the main 'quick print' screen of Cura. Here you can load and adjust 3D models, choose print profiles and upload files to the YouMagine library. Below you can see a quick overview about all the items in the interface. Later on in this chapter they will be explained in more depth

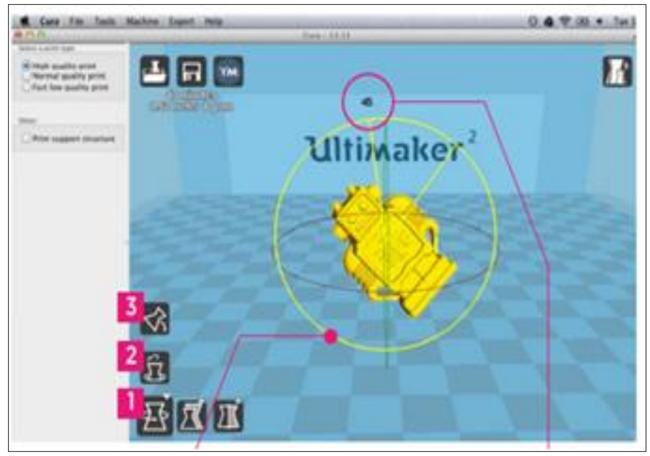


1. Menu bar In this bar you can change settings, machines and profiles.
2. Make a selection in 3 different quick print profiles.
3. The option to print with support structure.
4. A button which gives you the opportunity to load objects.
5. With this button you can save prepared files to your Printer SD-card.
6. Through this button you can share 3D files on YouMagine.com.
7. A prepared model can be viewed in other modes to check it's printpath.
8. The option to change the rotation of the object you like to print.
9. The option to change the Scale of the object you like to print.
10. The options to Mirror the model you like to print.
11. The model you have loaded through the load file button.
12. This is a visualization of the print area of your Printer.
13. The grey squares in the build area are the no go zones. In your Printer these are the metal clips were you can't print.

CURA Slicing Modes:

Adjusting Object

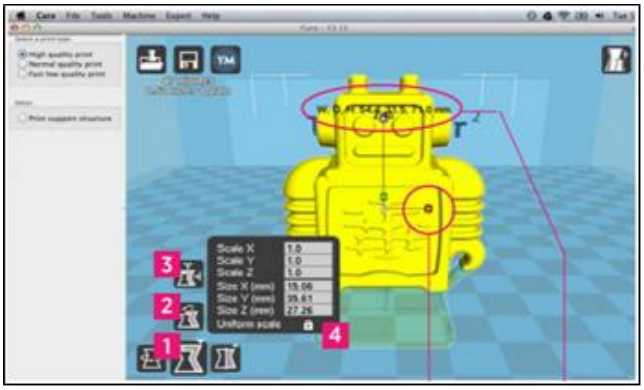
Rotating your object



The left icon on the bottom of the 3D interface is the rotation button [1]. When you select and click it, you can rotate the model over its XYZ axis. You see also more functions when you have selected the rotation button. the top icon's action, lays your model flat [3] on the surface, to make sure your model is well attached to the build plate while printing. The second icon resets [2] the 3D models rotation. By click-select one of the 3 orientation circles you adjust the rotation of the model. The rotation degree appears in the number around the model. When rotating and clicking shift you rotate per degree otherwise it's per 15 degrees.

Scaling your object

The second left icon on the bottom of the 3D interface is the scaling button [1]. When you select and click it, you can scale the model in the XYZ direction. The top icon that appears has the function to scale your object to max size [3] for your printer. The icon above resets [2] the 3D models rotation. By unlocking [4] the lock you have the possiblity to scale the object in each particular dimension. By select and slide one of the 3 scaling squares you adjust the uniform scale of the model. The amount of scaling and size dimensions appears in the number next to the model.

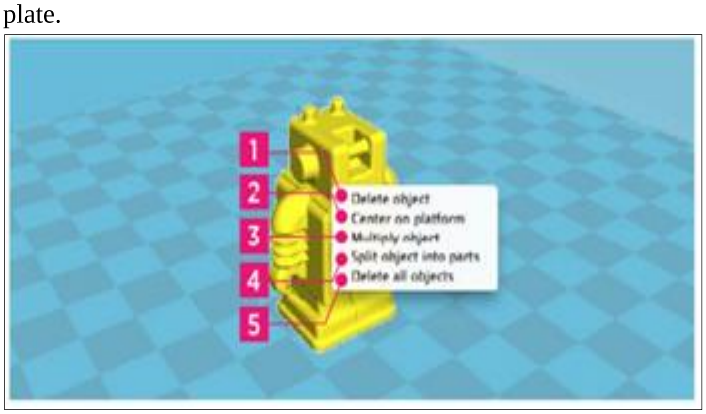


Mirror your object

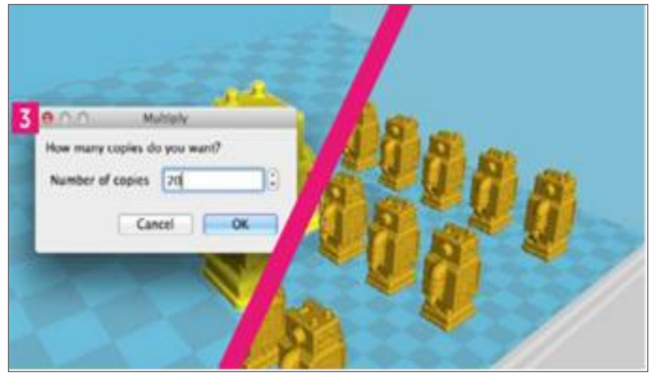
The third left icon on the bottom of the 3D interface is the mirror button [1]. When you select on the mirroring functions you can flip you model with the other buttons above [1] in the: - X dimension [2] - Y dimension [3] - Z dimension [4]

Right mouse click on your model.

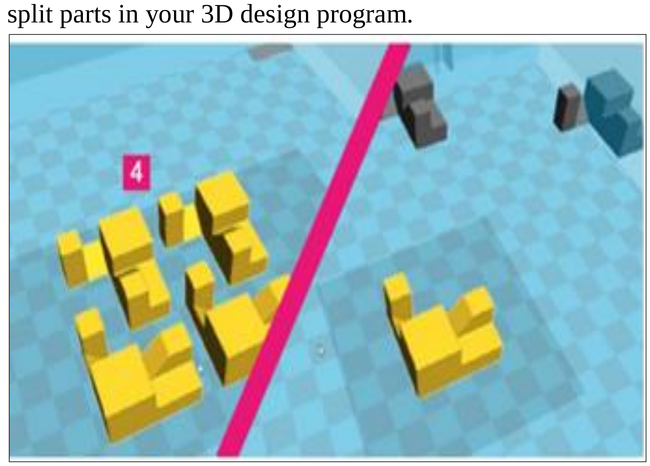
When you select your model and give a right mouse click on it you get some more functions. With Delete object [1] [5] you remove your model(s) from cura. Center on platform [2] positions your model in the center of the build plate.



With multiply object [3] you can decide to make more of the selected object. If you number is to much for the amount of space on your build volume cura will place it automatically it on maximum quantity.



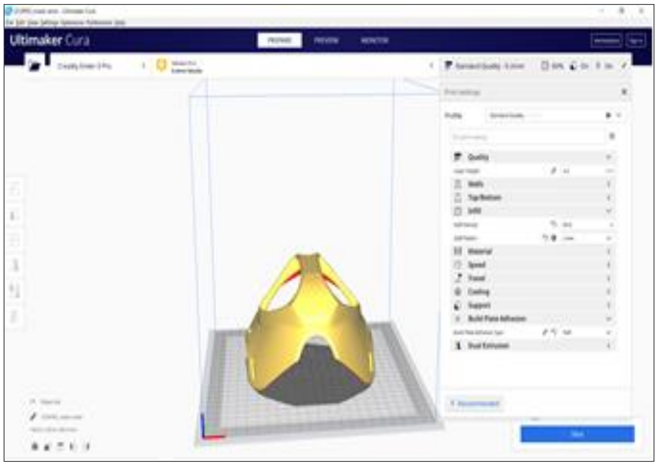
There is also a function Split object into parts [4], This function is used to split an model file which contains multiple parts into separate parts to be printed one at a time. For example, if you downloaded an STL file that contains 4 parts of a puzzle, you can split it into 4 separate parts for printing. For now it works very slowly and you might need some patience to split objects, for now we advise your to split parts in your 3D design program.



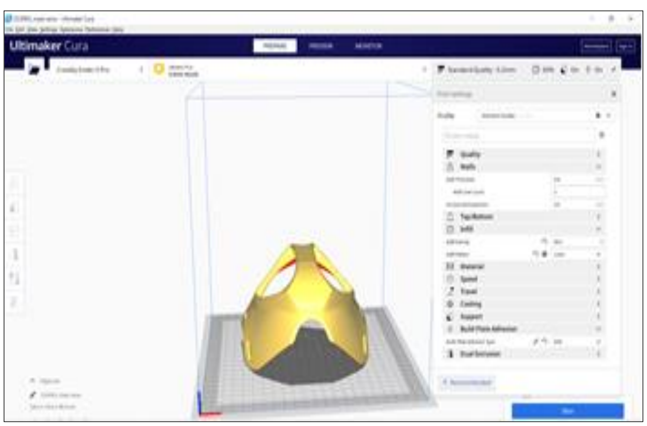
Quality Layer height:

One of the most frequently changed settings. It is the thickness of one printed layer in millimeters. With a thinner layer height, you can increase the quality of the print, leading to a smoother surface and more detail visible in the Z-direction (height) of the model. On the other hand, by using thicker layers you can decrease the print time substantially

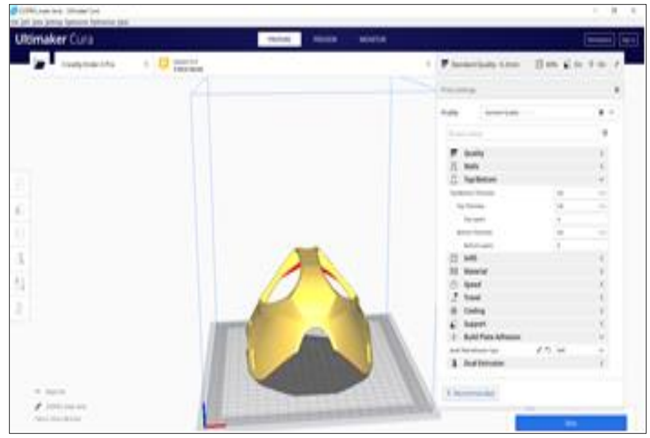
|| Volume 5 || Issue 12 || April 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH & ENGINEERING TRENDS
Multidisciplinary Journal
Double-Blind Peer Reviewed Refereed Open Access International Journal



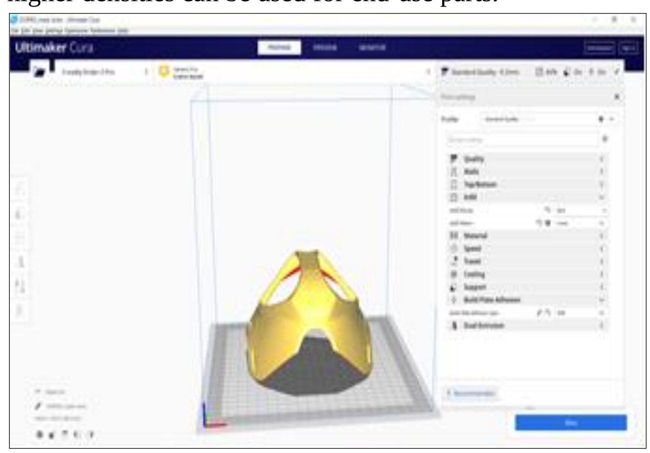
Shell Wall Thickness:
Number of perimeters compounding the wall of the model. BCN3D Cura rounds the wall thickness to a multiplication of the line width. In general, a wall thickness of two or three times the line width is sufficient. A higher value will create a sturdier model and decreases the chance of leaks, while a lower value can significantly decrease the print time and filament costs. Instead of setting a thickness in millimeter of the walls, you can also set a number of walls. When you set the wall line count, the wall thickness is calculated and will grey out.



Shell Top/Bottom Thickness: With the top/bottom thickness you can set the thickness of the solidly printed top and bottom layers of the print. A higher value ensures all gaps on the top and bottom layers are closed completely. However, this can also increase the print time and amount of filament used. It is advised to always use a multiple of the layer height for the thickness of the top and bottom.

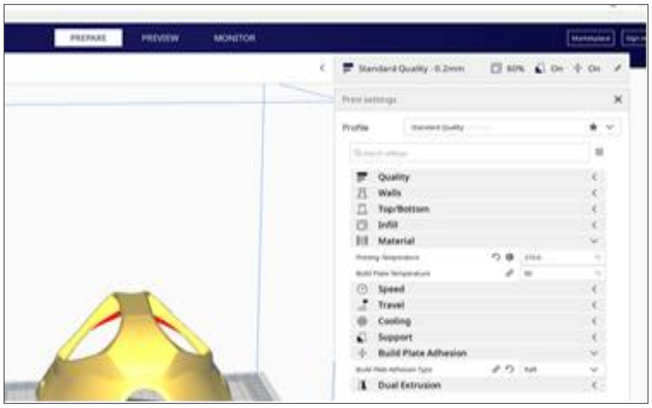


Infill Density: The infill density defines the amount of plastic used on the inside of the print. A higher infill density means that there is more plastic on the inside of your print, leading to a stronger object. An infill density around 20% is used for models with a visual purpose, higher densities can be used for end-use parts.



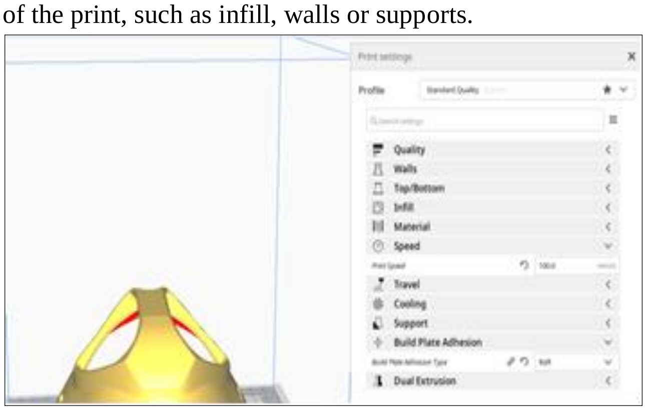
Material
Printing Temperature: This refers to the temperature of the nozzle while printing, including the adapted extrusion rate. Each printing profile has a slightly different printing temperature to create the best print result.
Build plate Temperature: This setting defines the heated bed temperature during the printing process. Each material has an ideal build plate temperature, which is set here.
Flow: It refers to the volume of plastic extruded. The automatically calculated value (100%) represents the theoretical amount of extruder plastic based on the filament diameter and feeder of the printer.

|| Volume 5 || Issue 12 || April 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH & ENGINEERING TRENDS
Multidisciplinary Journal
Double-Blind Peer Reviewed Refereed Open Access International Journal



Speed

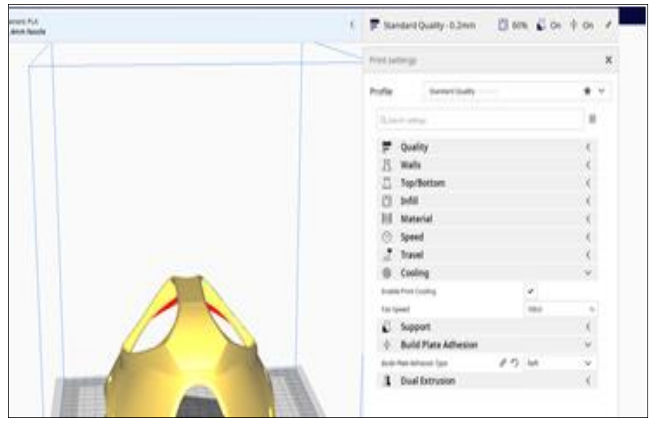
Print speed: The print speed defines the speed (in mm/s) at which the tool head moves while printing. Based on this setting, BCN3D Cura calculates the extrusion flow. The print speed can be visualized per feature in the Layer view -> Feedrate. A higher print speed will lead to a shorter print time. Keep in mind that increasing the print speed means that you may have to increase the temperature as well to ensure the filament is properly melted. Although you can choose one overall print speed for the complete print, it is also possible to use different print speeds for specific parts of the print, such as infill, walls or supports.



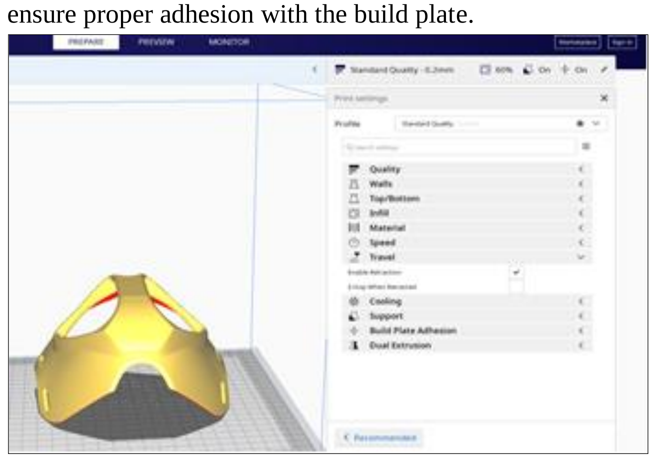
Cooling

Enable Print Cooling:

With this setting you can enable or disable the print head fans during printing. The print head fans ensure that the material is cooled properly before the next layer is printed. For layers with a short layer time, and those with bridges/ overhangs, cooling will increase the print quality. However, cooling may affect the layer adhesion of certain technical materials, such as ABS, causing layer cracks or mechanical defects. For this reason, some profiles set the cooling fans off. BCN3D Cura will automatically regulate the cooling fan speed depending on the layer time, geometry o height of the model.



Travel speed: This is the speed at which the print head moves when it's not extruding. A higher travel speed decreases the chance of filament oozing from the nozzle, resulting in a cleaner object. However, higher speed could also cause the nozzle to hit a previously printed part, which may damage the print due to the heated nozzle. This can be prevented by using Z-hop when retracting. The travel speed for the initial layer differs from the rest of the print to ensure proper adhesion with the build plate.



Support

Generate Support: Some models have overhanging parts, which means that parts of the model float mid-air when you would print the model. In this case, you must print a support structure under the model to prevent the plastic from falling down. This can be achieved by enabling this setting.

Support Extruder: Thanks to the IDEX architecture, BCN3D printers are able to print support structures with a water-soluble material called PVA, instead of using the exact same material of the model. Chose the extruder responsible to print the supports.

It is also possible to specifically choose the extruder to print certain parts of the supports: specific parts of the supports:

• Support infill
• First layer support
• Support interface
• Support roof
• Support floor This can be useful to save material and money, since PVA is more expensive than most of the filaments.

Build Plate Adhesion

Build Plate adhesion type: First layer is the most important moment of the 3D Printing process. A good first layer decreases significantly the chance of failure. For this reasons, there are different strategies to ensure a proper adhesion of the deposited plastic onto the build plate:

Skirt: A skirt is a line printed around the object on the first layer, but not connected to the object. This helps prime the extrusion nozzle and can be an additional check for bed leveling before the print begins.

Brim: Brim adds a single layer flat area around the base of the model to prevent warping. The brim is connected to the model and makes the bottom surface area bigger. This increases the adhesion to the build plate and, in case of warping, the corners of the model are less likely to curl up because of the brim attached to it.

***Raft*:** A raft adds a thick grid with a roof between the model and the build plate. This can be useful when the bottom surface of a model is not completely flat or has little adhesion to the build plate. A raft ensures that the model will stick better to the build plate.

IV. CONCLUSION

The intention behind this research was to study different 3 D printing process as well as software's used in complete process. We have been successful in studying different-different 3D printing processes and their real time applications. We studied various CAM tools in 3D printing such as Cura, KISSlicer, Skeinforge, Slic3r, RepSnapper, Miracl Makerware. Initially we compared various CAM tools used in 3D printing, in terms of ease of usefulness, flexibility, accuracy of prints, finally we concluded that CURA is best slicing tool due to its ease of use and flexible setting commands.

REFERENCES

1. 3D Hubs: Trend Report August http://www.3dhubs.com/ trends, Amsterdam, The Netherlands, last accessed 2014- 08-27
2. Agarwala, M.; Jamalabad, J.; Langrana, N.; Safari, A.; Whalen, P.; Danforth, S.: Structural quality of parts pro- cessed by fused deposition, Rapid Prototyping Journal, 2(4), 1996, 4–19. http://dx.doi.org/10.1108/13552549610 732034
3. Armillotta, A.: Assessment of surface quality on textured FDM prototypes, Rapid Prototyping Journal, 12(1), 2006, 35–41. http://dx.doi.org/10.1108/13552540610637255.
4. Blender, Blender Homepage, http://www.blender.org/, 2014, last accessed 2014-12-10
5. Chee Kai, C.; Kah Fai, L.; Chu-Sing, L.: Rapid prototyp- ing: principles and applications in manufacturing, World Scientific Publishing Co. Inc., River Edge, NJ, USA., 2010, ISBN 9812778977
6. Crump, S.: Apparatus and method for creating three- dimensional objects, 1989, https://www.google.com/ patents/US5121329
7. Fastermann, P.: 3D-Druck/Rapid Prototyping. Springer, 2012, ISBN 978-3-642-29224-8
8. Grunert, J.: Text Test, http://www.thingiverse.com/thing: 383163, 2014, last accessed 2014-08-27
9. Grunert, J.: Precision Test, http://www.thingiverse.com/ thing:383165, 2014, last accessed 2014-08-27
10. Hsiao, W.: Overhang Test Print (Customizable), http:// www.thingiverse.com/thing:58218, 2013, last accessed 2014-08-27
11. Khalyfa, A.; Vogt, S.; Weisser, J.; Grimm, G.; Rechten- bach, A.; Meyer, W.; Schnabelrauch, M.: Development of a new calcium phosphate powder-binder system for the 3D printing of patient specific implants, Journal of Mate- rials Science:

Materials in Medicine, 18(5), 2007, 909–916. http://dx.doi.org/10.1007/s10856-006-0073-2.

12. KISSlicer.com: KISSlicer (Keep it Simple Slicer), http:// kisslicer.com/, last accessed 2014-08-27
13. Malone, E.; Rasa, K.; Cohen, D.; Isaacson, T.; Lashley, H.; Lipson, H.: Freeform fabrication of zinc-air batter- ies and electromechanical assemblies, Rapid Prototyping Journal, 10(1), 2004, 58–69. http://dx.doi.org/10.1108/ 13552540410512543.
14. Makerbot Industries LLC: MakerBot 3D Printers 3D Printing, http://www.makerbot.com, last accessed 2014- 08-27