## 3D printing software

### **Introduction**

The software used for 3D printing has one basic task, which is to convert a file containing the model to G-codes. While some software is developed to work with specific 3D printer models, many software packages can be used to simply generate the G-codes. These G-codes can then be sent to a company manufactured or self-assembled 3D printer through various means.

### **Comparison**

A comparison of the most widely employed slicers was done to determine the best choice. While each application has its own benefits making it ideal to be used in some cases, the comparison below will provide a general idea of the best suited application. It should be noted that print quality also depends on factors such as retraction distance and speeds. For beginners, there are all in one products like MakerBot which can be used to print selected materials with ease.

|  |  |  |
| --- | --- | --- |
| **Cura** | **Simplify3D** | **Slic3R** |
| Slower processing of segmented layers | Nearly twice the speed of Cura | Around 10 times slower than Simplify3D making it the slowest choice |
| Easy to configure and print | Settings need to be adjusted but it provides more optimization to match user requirements | Can be configured for quick as well as professional use |
| Supports multiple programs and file types | Supports widely used file types | Prusa Slic3R 2.0 supports .3mf file type but other versions don’t |
| No need for remembering axis for rotation | No need for remembering axis for rotation | Needs to remember axis for rotation |

### **Prusa Slic3R**

For the purposes of this research, the Prusa Slic3R seems to be the best alternative. The steps for printing a model are given below:

1. Install and open Prusa Slic3R 2.0
2. Go to 3D mode and click on “add object”. The object cannot be added in layer mode. Users can use Ctrl+c and Ctrl+v to copy and paste respectively.
3. Use corner arrows to scale the object uniformly. Edge arrows can be used for distortion
4. Adjust settings as required. The program has a large number of settings however, the basic settings are given below:
   1. Print Quality – It depends on the number of layers which is determined by the thickness of each layer. For general purposes, layer size of 0.15mm provides good quality
   2. Filament – Different materials and plastics are supported as the filament type. If the brand is not present in the options then user can select “generic PLA”
   3. Printer – Different types and brands of printers are supported directly. For self-assembled or modified printers, the user can also configure the printer
   4. Supports – These are required to support the main structure and are removed afterwards using different processes. To generate supports, right click on the model and choose “add support enforcers”. To block support right click on the model and select “add support blocker”, choose the shape and size of blocker and position it using the move tool. Dual nozzles can help with supports which can dissolve in certain medium or temperature without affecting the part making it very easy to remove them. There are two general methods of defining supports:
      1. Generate supports for the entire body using “everywhere” option. Then place blockers to remove supports within regions where they are not required
      2. Generate supports only in regions where they are needed. This method generates more supports than suggested by the algorithm. However, the model is more consistent even though it uses more amount of plastic
   5. Infill – This is the amount of plastic present inside the outer structure and is important for providing strength to the printed part. The amount of infill depends on the model type along with its intended use
5. Click on “Slice model” and then select “export G-Code to SD card” for sending it to the printer