3D Slicing Basics

Basic vocabulary

Slicing is the term for preparing a 3d Model for printing on the 3d Printer- it generates a G-code that instructs the printer on what to print and how to print it, layer by layer.

Filament is the material used to print your models- the printers in the Makerspace use PLA, but there are many other sorts of filament that have different properties.

G-code is the computer code that gives instructions to the 3D printer, telling it how to print your model.

Infill determines the percentage of filler in the hollow space inside your 3D model when it prints

Supports are extra structures outside your model that allow it to print overhanging parts of the model

Build Plate is the plate that the 3D printer builds the model on. **Build plate adhesion** refers to additional structures that help keep your model fixed and stable on the build plate during printing.

Finishing refers to anything you need to do to clean up your model after printing. This may include removing supports and build plate adhesion structures, removing stringing or other unwanted bits of filament that may be left behind from printing, or sanding/ smoothing rough surfaces.

3D Printing talking points

- 3D printing is a hobby involves a lot of trial and error to get successful prints- expect to have to tweak and adjust settings in order to get a good result.
- Not all 3D models are going to be suitable for printing (on Thingiverse.com or wherever you find your model or create it). This may be because of printer limitations (not all printers are created equal), model design, or just practical questions like how much time it will take to print.
- Think about how your model will sit to maximize contact with the build plate and minimize overhanging parts that will require supports.
- Always save your file 'as a project' after you slice and save the G-code. This will allow you to come back and make changes to your settings and fine tune the print.

Choosing or creating a model to print- factors to consider

- **Size**: large models can be scaled down to fit, but fine details become more challenging to print as they get smaller.
- **Complexity**: the more *complex* a model is- lots of detail, lots of surfaces- the longer it will take to print.
- **Detail**: finer detail may require choosing a smaller layer height to print successfully, increasing print time.
- **Fragility**: models with small, thin, or otherwise fragile parts, especially if they need supports, will be harder to successfully print and may end up damaged in finishing.
- **Supports**: models that will need a lot of supports, especially if they are a lot of freestanding supports or a lot of supports that will be wrapped around the model, may be hard to clean up after printing. Supports also add time to printing.

- **Adhesion**: models often have an obvious ‘bottom’ that will sit flat on the build plate. If there is no obvious bottom, you’ll have to look for a way to orient the model to have the most contact with the build plate while allowing for the least needed support.

Settings to check/ choose each time you slice a model

Look at and make choices about these settings every time you slice a model. There are *many* other settings you can look at and adjust, but these are the basic and generally the most critical ones.

Layer Thickness:

- Layer thickness determines overall quality: - thinner layers mean higher resolution quality.
- Layer thickness should be set as a multiple of the nozzle diameter (ours is 0.4 mm).
    - 2.4 mm: Low resolution
    - 1.6 - 2.0 mm: Standard resolution
    - 1.2 mm: Fine resolution

Infill:

- Infill Density- determines the amount of filled versus empty space that is automatically generated inside the walls of the model. The higher the density, the heavier, sturdier the model will be, but the more filament and time will be needed to print. 7-20% is a good range.
- Infill Pattern- a grid pattern is functional and fast for most prints. For prints that require more strength and durability, or other special situations: Cura Guide to the Best Infill Patterns | All3DP

Material:

- Printing Temperature- determines how hot the print head will get; different filaments may have different temp ranges. The PLA we are currently using seems to work best at 220-230° Celsius.

Speed:

- Print Speed- sets the base speed for printing, 300 mm/s is a good speed for most prints. More complex prints might benefit from slower speeds.

Support:

Every layer printed *must* build on top of the layer below it. If your model has parts that will *not* be able to build on the layer beneath, you will need to use supports to get a successful print. Usually these are automatically generated, but some models have supports built into the design. *All settings for supports are decisions that will need to be made each time based on the specific needs of a given model.*

- **Generate Support**- this checkbox lets the slicer know whether you want automatically generated supports or not.
- **Support Structure**- determines the kind of structure generated for support. Experiment with both kinds to find the one that works best for your project.
    - Normal- vertical towers built up under supported areas.

    - Tree- organic, branching structure that 'grows' up under supported areas.
- **Support Overhang Angle**- sets the angle that will trigger supports to be generated- the lower the angle, the more areas that will generate supports. For most models needing supports, 65-80° is good, but this may differ based on the needs of a specific model.
- **Support Density**- determines how sturdy supports will be. Heavier supports will stand up stronger, but lighter supports will break off easier. 7-15% is usually good.
- **Support Z Distance**- sets the distance between the top or bottom of the support structure and its contact points on the model or build plate. The smaller the distance, the sturdier the support, but the harder it will be to remove after. For the top 0.5-0.8 mm is generally good, for the bottom 0.4-0.6 mm.
- **Support X/Y Distance**- sets the distance between the supports and model for any points of contact on the X/Y axes. 0.7-0.8 mm is usually good.

Build Plate Adhesion:

Determines what structure (if any) is printed to help the model adhere to the build plate during printing. Periodically we apply glue stick to the build plate to aid in adhesion.

- **None**- models with larger, connected, flat surfaces in contact with the build plate generally need no extra help with adhesion.
- **Brim**- generates a thin layer around the outer perimeter of the build plate contact points of the model, anchoring small points of contact in place during printing. Usually pretty easy to remove after printing.
- **Raft**- generates a heavier, solid layer as a foundation for the model that then prints on top of it. Helps models with minimal contact points or higher reaching, free-standing pieces to stay anchored. Uses more filament and adds substantial print time. It can be harder to remove than a brim.
    - **Raft Air Gap**- sets the distance between the top of the raft and the bottom of the model. Higher air gap makes the model easier to remove, but lower air gap makes a sturdier point of contact. Typically, an air gap of less than 0.4-0.5 mm will be difficult to remove.