Within this module, you will be introduced to 3D Print File Setup, also known as Slicing. Slicing is the process of applying settings and orienting your model so that the software can generate code to communicate with your printer. Setting up your slicer settings is crucial to a successful print, and to bringing your digital model to life. There are several popular slicers out there, but for this assignment, we will be focusing on two, Prusa Slicer and Ultimaker Cura. Whichever one you decide to use will work for this assignment and each has its own unique benefits. For future use, we recommend you get used to both.

Final Deliverable

One .GCODE file sliced for CR-10

Instructions

1. Installation

Download Cura: https://ultimaker.com/software/ultimaker-cura

Download PrusaSlicer: https://www.prusa3d.com/page/prusaslicer_424/

Follow the install instructions and open the selected software.

Drag in your 4”x 4” x 4” model from Assignment M3A1 or M3A2

2. Orientation

Following the THEY Rules, orient your object to have the HIGHEST amount of surface area touching the build plate while also REDUCING the amount of support required

3. Settings

Infill - The amount of structure within your walls

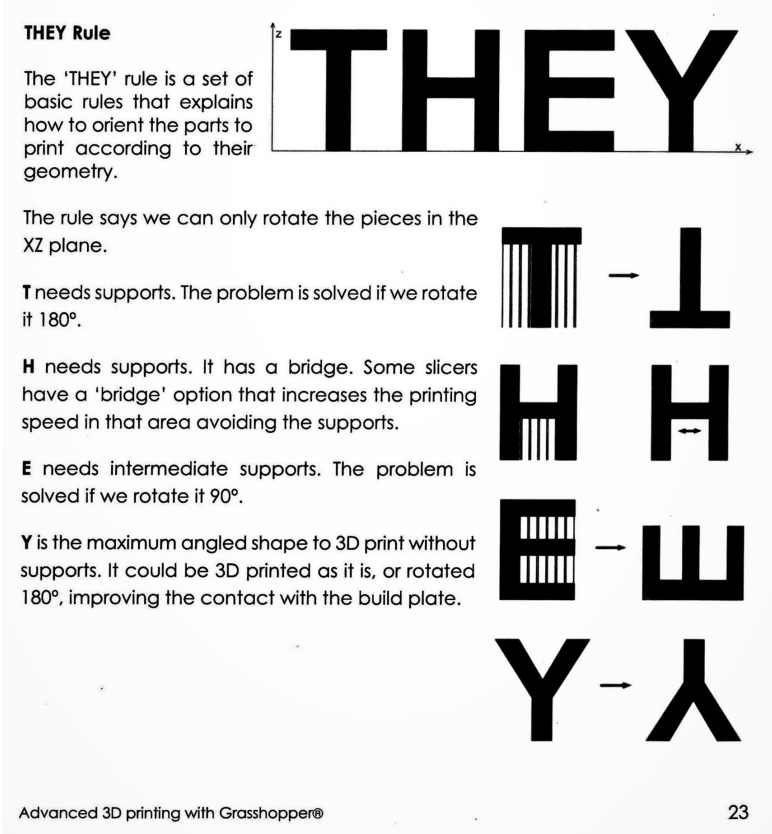
20% is standard but can go up or down depending on your requirements.

Speed - Typical printers can handle 60mm/s but can go up or down depending on the material or required finish quality

Layer Height- Determines finish quality typical is .2mm(Most important time saver setting)

Adhesion- Enable whenever possible

Support - Reduce usage as much as possible but enable to ensure print quality and stability



Source :Advanced 3D Printing with Grasshopper Clay and FDM- Diego Garcia Cuevas , Gianluca Pugliese