

# 3D PRINTING GUIDE

## CREATING YOUR 3D OBJECT

1. In order to print an object, you will need a .OBJ or .STL file
2. There are many open source objects online that people have already created, however you are strongly encouraged to try designing your own. The Autodesk suite of software is great for designing 3D objects
3. If you do not have any experience with 3D design I highly recommend starting with Tinkercad.com. Tinkercad offers free online software that is very intuitive to use and great for beginners

## SLICING YOUR 3D OBJECT FILE

1. In order to print a 3D object you will need to "slice" your .OBJ or .STL file and turn it into a .GCODE
2. This is done by using a slicing program like Ultimaker Cura
3. Open Ultimaker Cura
4. If it is the first time you are using Cura you will need to set up the printer
5. Click Add non-network printer then select the correct printer from the drop-down list, the majority of our printers are Creality Ender 3's
6. To import your file into Cura you can click either the folder icon in the top left or go to File>Open File
7. Select your file and it will be placed into Cura
8. With your object selected you can use any of the following 6 functions on the left side of the screen:

- **Move** – This allows you to reposition your object anywhere on the printing bed
- **Scale** – This allows you to resize your object
- **Rotate** – This allows you to rotate your object 360° on 3 axis
- **Mirror** – This allows you to create a mirror image of your object
- **Mesh Type** – This allows you to set the mesh type as normal, print as support, modify settings for overlaps, and not support overlaps
- **Support Blocker** – This allows you to select areas of your object where you do not want supports to be added



9. Using the bar on the top of the screen you can select the type of plastic you are printing (typically we use PLA) and the quality of the print.
10. The quality of print drop-down menu gives you options for:
  - **Profiles** – Determines the resolution of the print default=.2mm
  - **Infill (%)** - Determines how dense the interior of the object will be default=20
  - **Support** – Check this box to automatically add supports to your object (supports allow you to print objects with overhangs)
  - **Adhesion** – Check this box to add adhesion (helps object stick to base)
11. After you have the setting where you want them click on the Slice button in the bottom right corner
12. Once your file is sliced save it to a micro SD card

## PRINTING YOUR FILE

1. Make sure the printer bed and nozzle are free of any debris, if there is anything on the bed you can use a scrapper to scrape it off
2. Place the micro SD card into the slot on the front of the printer
3. Turn on the printer using the red button on the left
4. Push the round button then turn it to scroll down to "Print from SD" push the button to make your selection then scroll to your file and push again to select
5. The printer will start to heat up, once the nozzle and bed are heated it will start to print
6. Watch your print to make sure that the first few layers are sticking to the bed (If you are having trouble with your print not sticking then you may want to turn up the heat on the bed, or consider adding adhesion.)
7. If your object looks like it is sticking to the bed and printing correctly you can leave it to finish.

## 3D PRINTER MENU OPTIONS:

### PREPARE

**DISABLE STEPPERS** – allows the bed to be moved freely by hand

**AUTO HOME** – returns the nozzle to its home position

**PREHEAT PLA** – preheats the nozzle and bed to PLA temps.

**PREHEAT ABS** – preheats the nozzle and bed to ABS temps.

**COOLDOWN** – returns the nozzle and bed to room temp

### MOVE AXIS

#### MOVE 10MM

**MOVE X** – moves X axis in 10mm intervals

**MOVE Y** – moves Y axis in 10mm Intervals

#### MOVE 1MM

**MOVE X** – moves X axis in 1mm intervals

**MOVE Y** – moves Y axis in 1mm intervals

**MOVE Z** – moves Z axis in 1mm intervals

**EXTRUDER** – extrudes in 1mm intervals

### CONTROL

#### TEMPERATURE

**NOZZLE** – set nozzle temp

**BED** – set bed temp

**FAN SPEED** – set fan speed

**PRINT FROM SD** – prints file from SD

**CHANGE SD** – loads new SD card