3D PRINTING

SOFTWARE GUIDE

CREATING A 3D OBJECT

1. In order to print a 3D object you will need a 3D object file (.obj or .stl)
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

WATCH TINKERCAD TUTORIAL:

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** (cura can be downloaded for free at www.ultimaker.com/software
4. If it is the first time you are using Cura you will need to set up the printer:

   Click Add non-network printer then select the correct printer from the drop-down list, the majority of our printers are **Creality Ender 3 V3 KE's** the large printer is the **CR-M4**

5. To import your file into Cura you can click either the folder icon in the top left or go to File>Open File
6. Select your file and it will be placed into Cura
7. With your object selected you can use any of the following 6 functions on the left side of the screen:

   - **Move** – Allows you to reposition your object anywhere on the printing bed
   - **Scale** – Allows you to resize your object
   - **Rotate** – Allows you to rotate your object 360° on 3 axis
   - **Mirror** – Allows you to create a mirror image of your object
   - **Mesh Type** – Allows you to set the mesh type
   - **Support Blocker** – Allows you to select areas where you do not want supports

8. 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.
9. 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)

10. After you have the settings where you want them click on the Slice button in the bottom right corner
11. Once your file is sliced save it to a USB