# 3D Printing

## What is 3D printing

- 3D printing is the process of depositing melted plastic in layers to manufacture a part.
- This process allows for rapid prototyping of complex designs and is a great choice for manufacturing your project.
- 3D printing supports a wide range of plastics, the most common including **PLA**, **ABS**, **PETG.**

**Prusa MK4**

## Steps to making a printed part

**Printer Fillament**

- You will first need to produce or find a 3D model of what you want to make.
- A great online library of 3D printable models can be found at **thingiverse.com**
- For making your own 3D models you will need to use a CAD software, like Fusion 360 or Solidworks.
- If your only making simple 3D models, the browser based CAD editor **tinkercad.com** is a great place to start.
- Once you have a 3D model, you will need to **slice** that model into a **GCODE** file, instructions that a 3D printer can follow to make your part.
- I would recommend using **Prusa Slicer** (this software will have to be downloaded).
- Finally, transfer your **GCODE** file onto an **SD card**, insert it into the printer, select your file and you will soon have brought your model into reality.

*PTO to see more details on **Tinkercad**, **Prusa Slicer** and the **printing process***

# How to use Tinkercad

- Go to **tinkercad.com.**
- login with google or make a new account with your email.
- Press the **create new project/part** button to take you to the editor.
- In the editor, drag and drop shapes into the workspace from the shapes browser.
- Resize the shape(s) to a desired size with the mouse or by entering dimensions.
- The whole idea of **tinkercad** is that you group shapes to make more complex models
- Shapes can be made **negative / hollow** to make cavities when grouped.
- Your model can be **exported** to .stl for use by a printer slicer software.

# How to use Prusa Slicer

- Prusa slicer can be downloaded from **prusa3d.com i**n the software tab, download and install to get started. On first launch you will have to apply some settings, leave everything default, you can change it later if needed.
- Once in the workspace drag and drop your **.stl** file into it.
- The default **material** the slicer uses is **PLA**.
- You can configure **speed**, **supports** for overhangs, build plate adhesion, etc.. in advanced mode. Any option you can think of probably exists, feel free to explore the slicer.
- Press slice button in bottom right corner.
- Then save your **.gcode** file to your **SD card.**
- Your now ready to print your model.

# Printing process and common issues

- Turn on the **3D printer** and Insert the **SD card.**
- Press the knob to enter the **main menu**, select the **SD card** and choose your file.
- Printer will then start **nozzle/bed heating** and **calibration**.
- Then the print will start.
- Press the **X** button if there is a problem and the printer will stop.
- Sometimes the slicer produces a **.bgcode** file instead of a **.gcode** file and our printers cannot read this, change the slicer settings to use **.gcode** instead**.**

*Worksheet produced by Lukas Hastings for use by HackSussex*
*https://github.com/supersand21/Robotics-Workshop*