MODEL

3D PRINTING CHEAT SHEET

Download a model from Thingiverse

3D scan and clean up in Meshmixer

OR

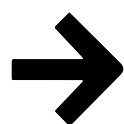Design an object in a computer aided design (CAD) program like Fusion360

Output an .stl or .obj file

KEEP IN MIND:

- *Object orientation matters*
- *Minimise overhang, and/or use appropriate supports*
- *Consider your choice of fabrication method – 3D printing is not suitable for all projects*
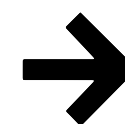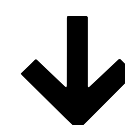
SLICE

IMPORT YOUR MODEL →

Download Cura from the Ultimaker website

Choose your printer based on availability or your project constraints (Ultimaker 2+, Ultimaker 2 Extended+ or Ultimaker 3).

Use the default MakerSpace print profiles, or customise the settings for your project ↓

SLICE
check weight and time of print

↓

SAVE
to SD card or USB (depending on chosen printer)

IMPORTANT SETTINGS

SETTING	DEFAULT	ALT. RANGE
layer height	0.12 mm	0.08 – 0.3 mm
wall thickness	0.8 mm	0.4 – 1.5 mm
infill	20%	15% – 50%
supports	use for overhangs or for steep angles	
build plate adhesion	brim	skirt

PRINT

troubleshooting

POOR ADHESION TO BUILD PLATE

- Footprint matters – make sure you have enough surface area contacting the plate. If too small, use a bigger brim. Consider object orientation when positioning on plate.

- Plate may require cleaning (see staff for reapplication of the build plate adhesive, Magigoo)

- Check your build plate levelling

UNDER- or NO EXTRUSION

- Reload filament (check for worn area at the feeder)

- Try unblocking the nozzle

- If you have tried all the above, the nozzle may be irreparably clogged. See MakerSpace staff for help changing out the nozzle.

see the Ultimaker website for tutorials on just about every topic
https://ultimaker.com/en/resources/

→ **LOAD YOUR FILAMENT** check you aren't using someone's personal filament

→ **LEVEL YOUR PLATE**

→ **PRINT** finetune levelling if required

→ **WATCH FIRST LAYER** fill out 3D printing log while you wait (if using MakerSpace or Class filament)