

Instructions for using the CRR10: Creality 3D - Our Trustworthy 3D printer

FABLAB BRIGHTON 2018



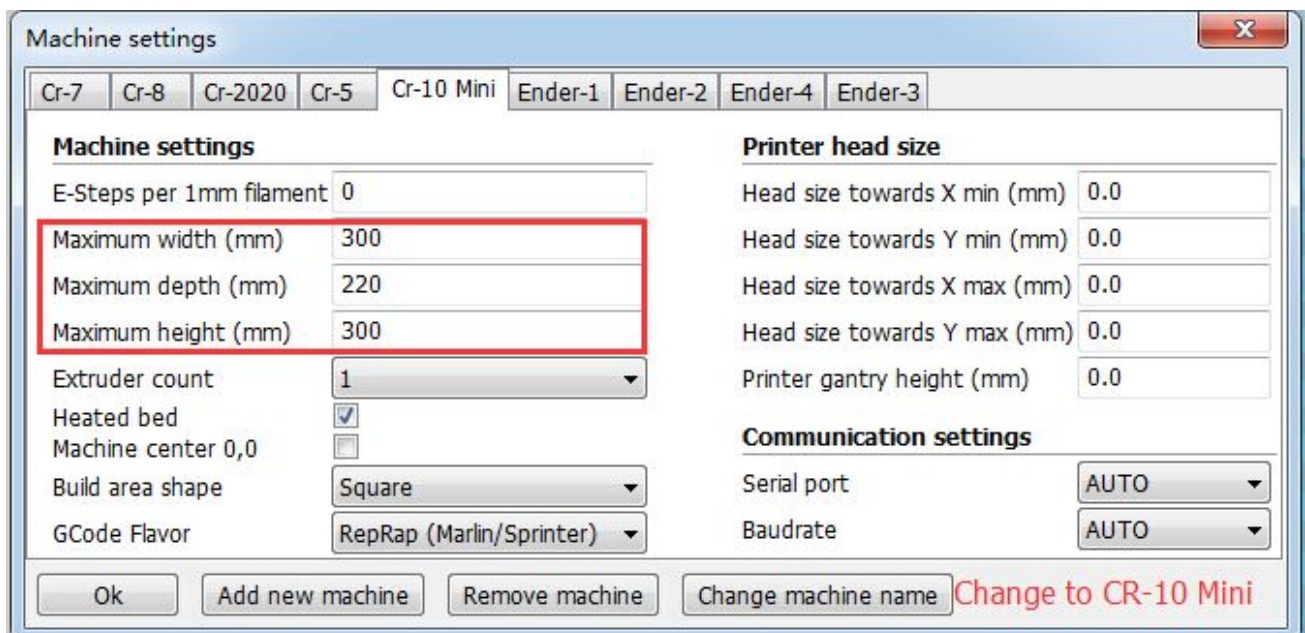
Installing Curamaker software

The first step is to download the software for this machine. The one we've been using is called Curamaker, by Ultimaker, and download it from this site:


















<https://ultimaker.com/en/products/ultimaker-cura-software>

Machine settings

Here are the machine settings required for Cura, specifically for our machine which is like the CR10...we've called it the CR10 mini because the bed size is slightly smaller. We need to change the bed size as shown below:

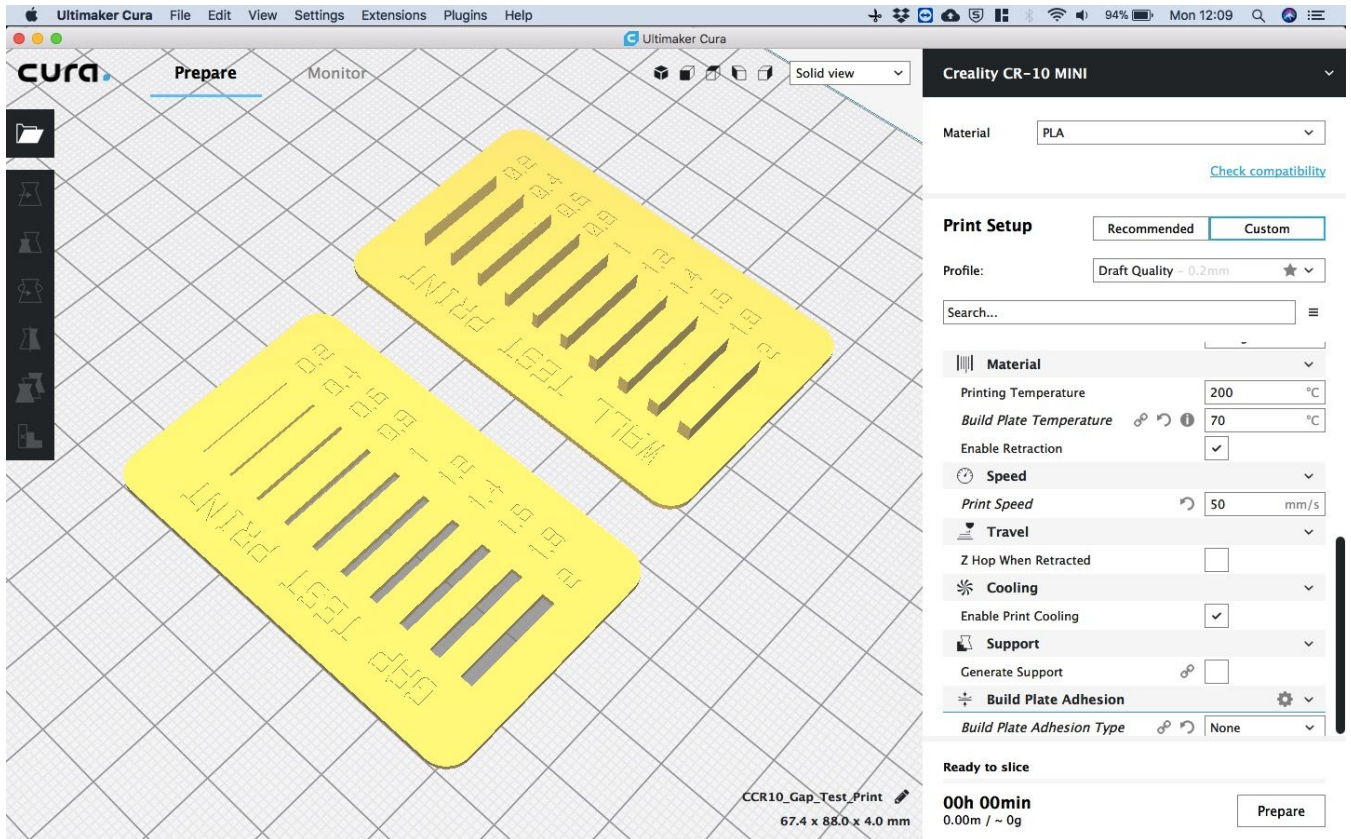


And then the main thing to change in the Custom print setup is the bed temperature...for this machine **THIS MUST BE 70 deg C**. The other big consideration is whether you want support material or not. Here are the basic settings we used to start with:

| | | |
|---|--|------|
|  Quality | | ▼ |
| Layer Height |  0.2 | mm |
|  Shell | | ▼ |
| Wall Thickness | 0.8 | mm |
| Wall Line Count | 2 | |
| Top/Bottom Thickness | 0.6 | mm |
| Top Thickness | 0.6 | mm |
| Top Layers | 3 | |
| Bottom Thickness | 0.6 | mm |
| Bottom Layers | 3 | |
| Horizontal Expansion | 0 | mm |
|  Infill | | ▼ |
| Infill Density | 20 | % |
| Infill Pattern | Triangles | ▼ |
|  Material | | ▼ |
| Printing Temperature | 200 | °C |
| <i>Build Plate Temperature</i> |    70 | °C |
| Enable Retraction | <input checked="" type="checkbox"/> | |
|  Speed | | ▼ |
| <i>Print Speed</i> |  50 | mm/s |
|  Travel | | ▼ |
| Z Hop When Retracted | <input type="checkbox"/> | |
|  Cooling | | ▼ |
| Enable Print Cooling | <input checked="" type="checkbox"/> | |
|  Support | | ▼ |
| Generate Support |  <input type="checkbox"/> | |
|  Build Plate Adhesion | | ▼ |
| <i>Build Plate Adhesion Type</i> |   None | ▼ |

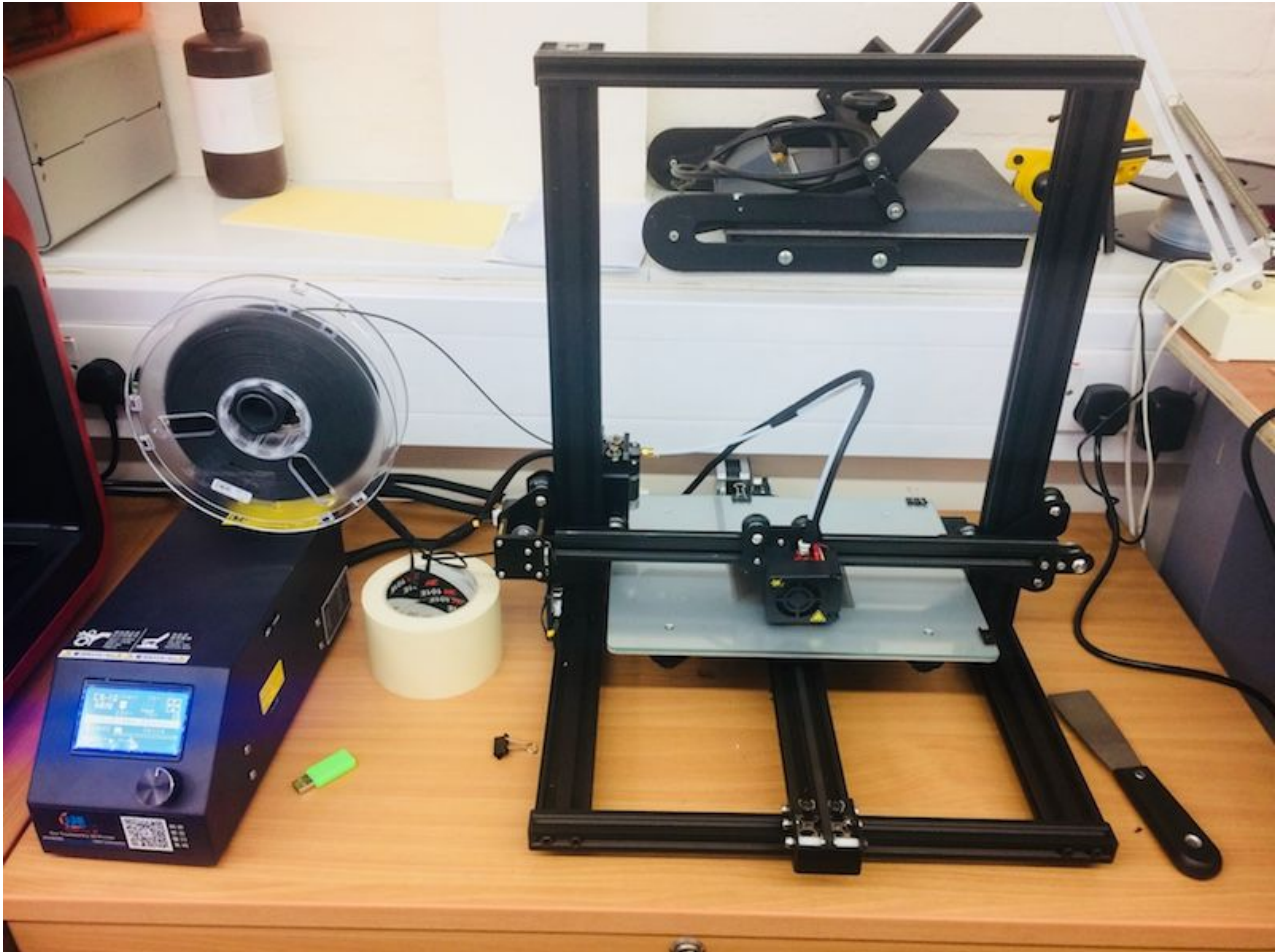
Loading your file into the software

File > Open Files...then search for your CAD model (typically in .stl format) and then hit OPEN. You can then re-orient, scale, mirror, position etc your model on the bed. It will be then ready to prepare the model by slicing it up and this will be turned into G-Code by hitting the SLICE button (bottom right). Here's how it will look:

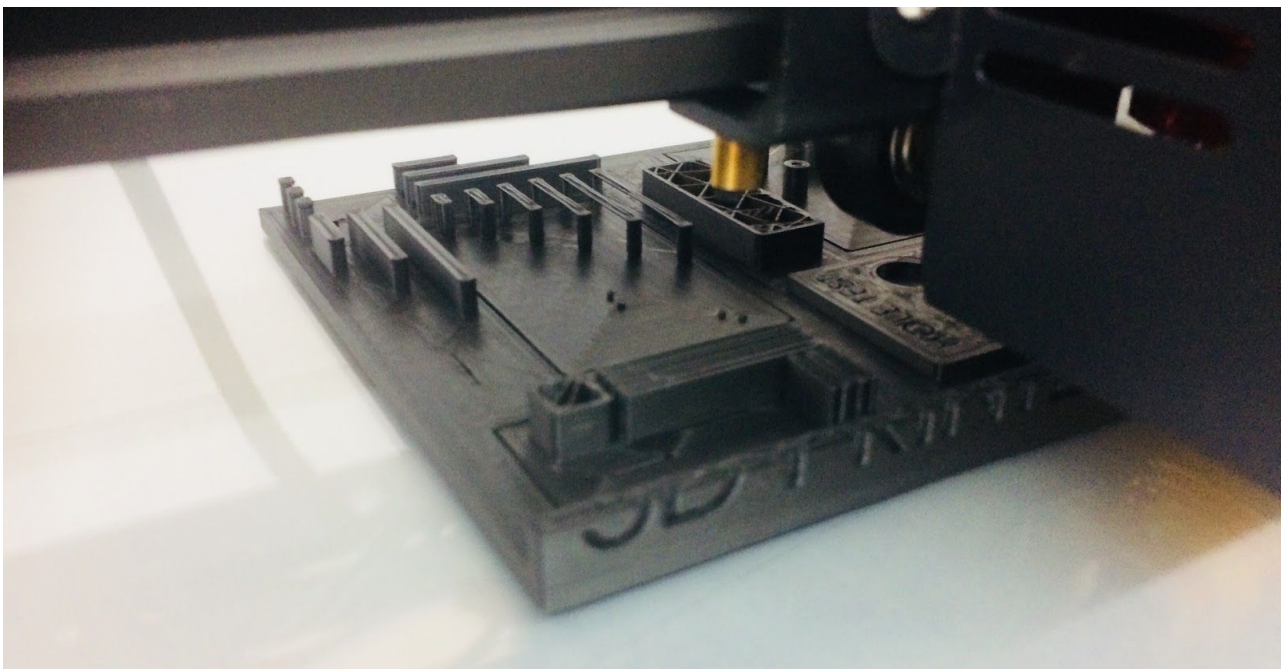


Once this is generated, Cura has a really nice feature, where if you plug in your USB stick, it will ask you if you want to save the G-Code directly to your USB device (button on bottom right again). Note that you can ONLY use the special FLUORESCENT GREEN USB stick. This has a small SD card embedded within it that can then be plugged directly into the 3D printer.

The fluoro green USB stick and 3D printer setup can be seen below. You can also see the LCD on the interface (left), which is where you press the dial, scroll down to PRINT FROM SD CARD, then find your G-Code file and then press the dial again to start the printing. Note that it will take a few minutes to warm up the printer bed before the actual printing will start.




And here it is in action.



Here's a short video showing the process (double click on the image below to play the video).

YouTube ^{GB}



3D printing with CRR 10 creality 3D - our trustworthy 3D printer
Unlisted

Printing calibration test models

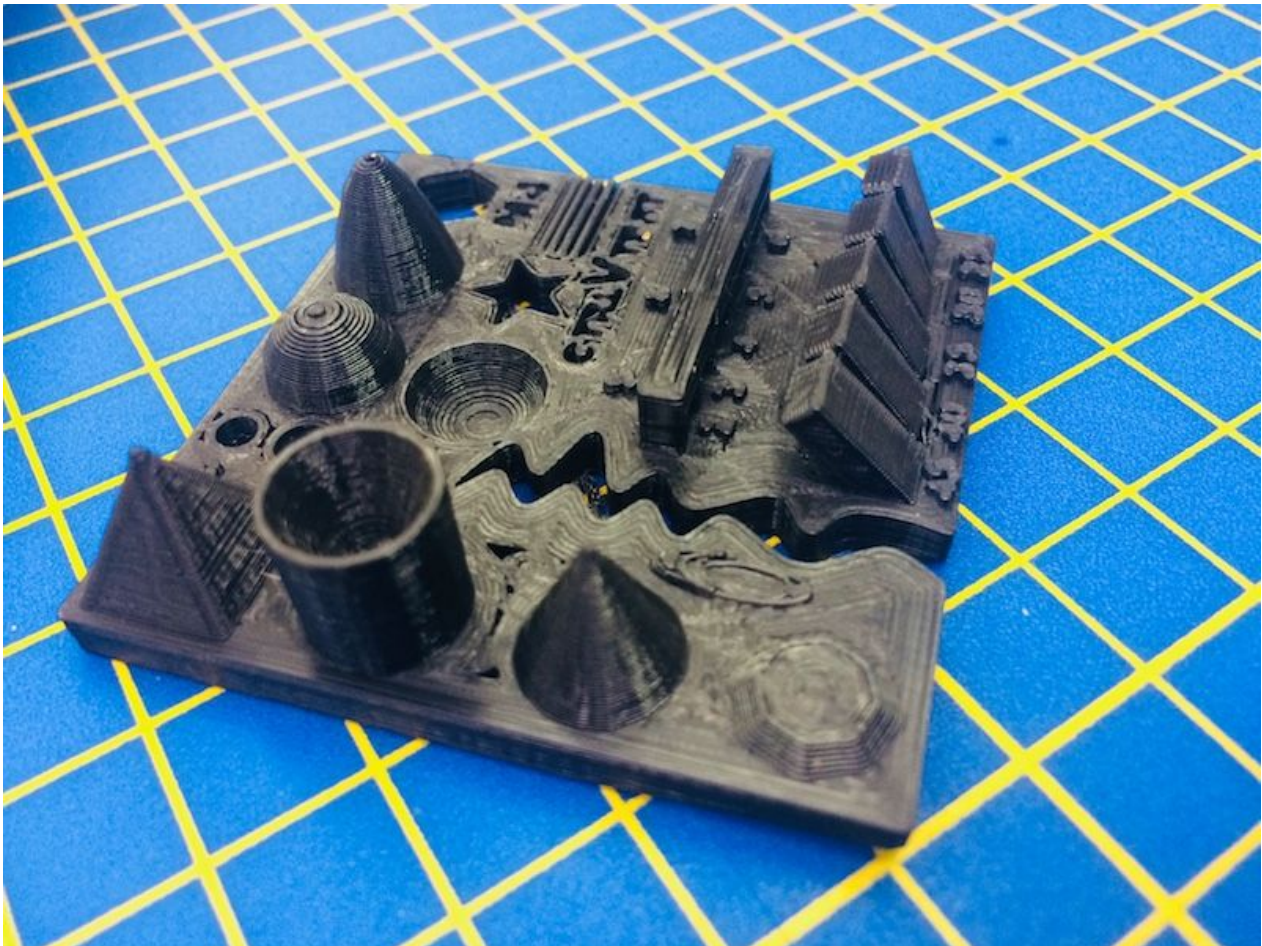
We then printed a series of test models to check the capabilities, design rules and limitations of this machine. And what a machine! We were very impressed by this little beauty. So easy to use, so impressive with reliability and repeatability (so far), and all for only about £300. Here are the three models we adapted from thingiverse. In particular these were good at testing: overhangs at various angles, unsupported beams, wall thicknesses, slot/gap thicknesses, text debossing, etc.

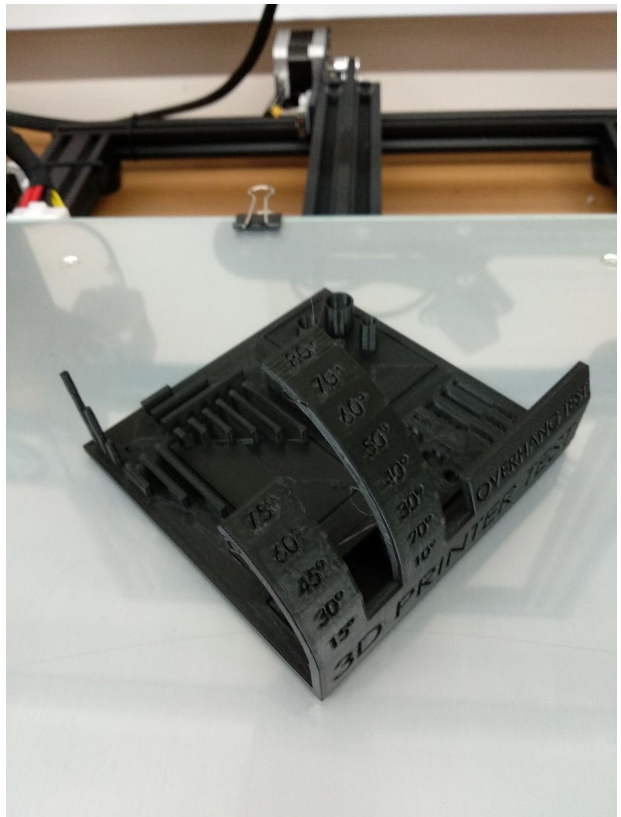
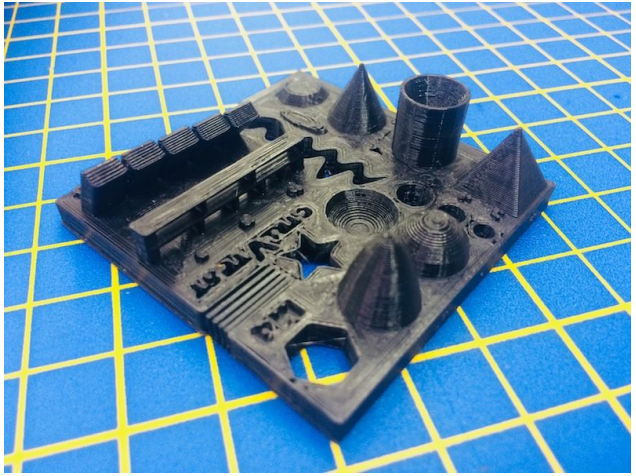
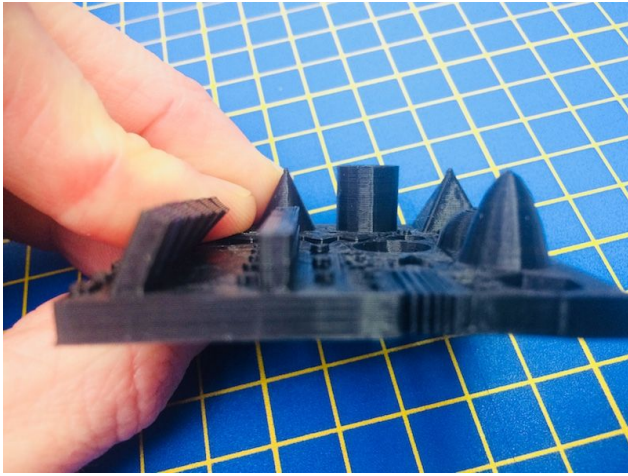
To summarise key outcomes:

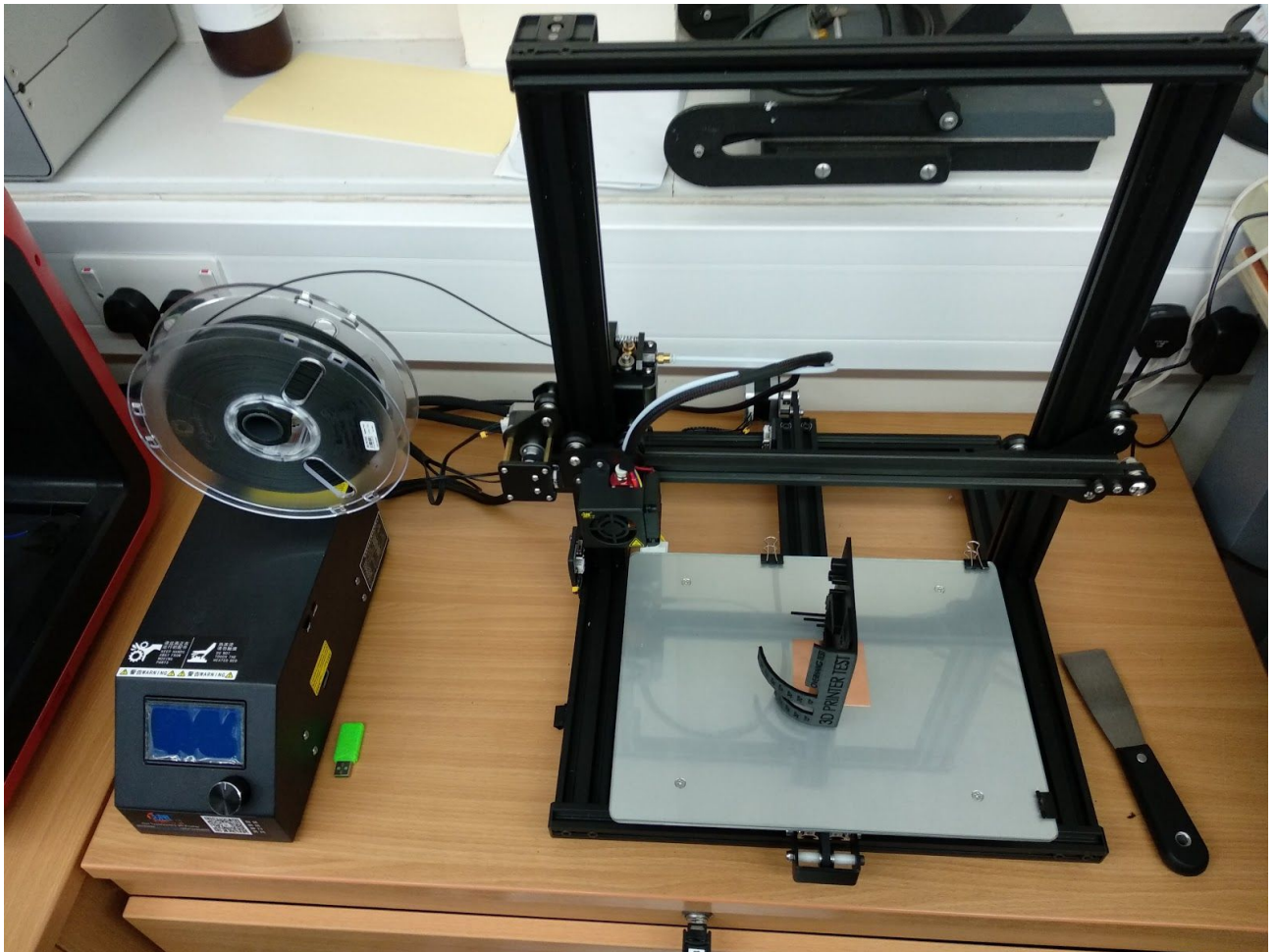
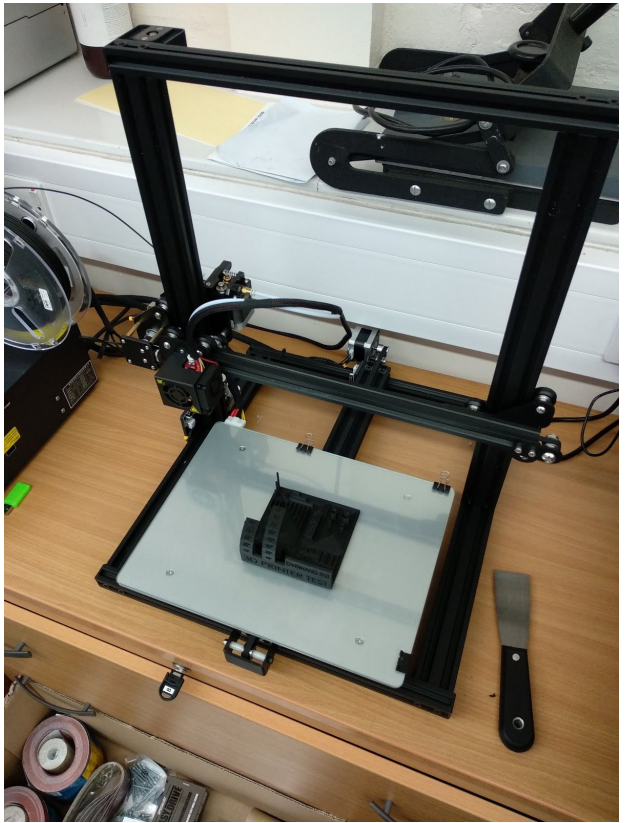
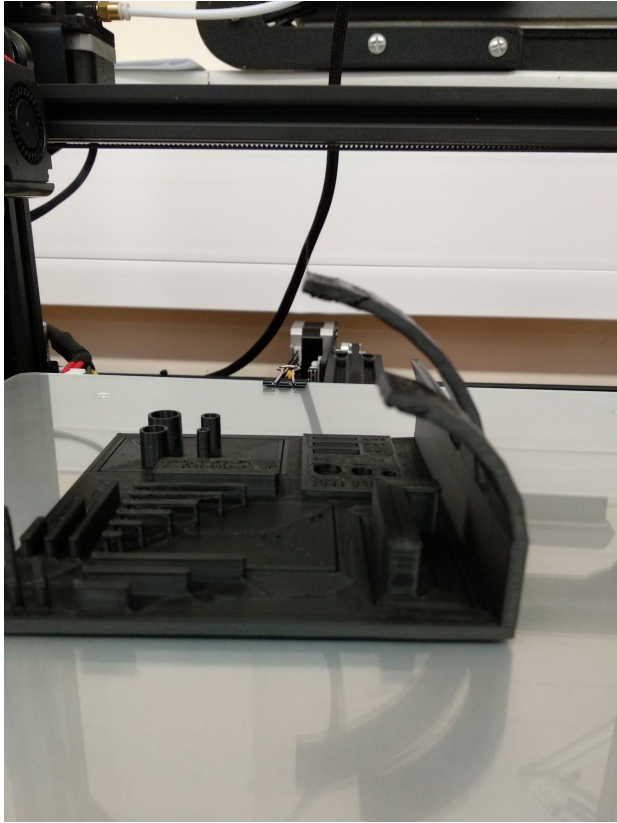
- We could only print standalone walls down to 0.6 mm wall thickness. 0.4 mm and 0.2 mm did not print at all. Note we didn't try 0.5 mm, but I suspect this would work.
- Walls tended to be larger than drawn (mean increase 0.03 mm)
- Gaps tended to be smaller than drawn (mean decrease 0.19 mm)
- Overhangs were limited to XX degrees from the vertical.



And here are a few images of the parts themselves:







Gap and wall measurement data

We then measured the actual gap and wall dimensions with our trusty calipers. Here is the data.



| | SPECIFIED | MEASURED (mm) | | |
|---|-----------|-----------------|------|-----------|
|  | 0.2 | X | n=10 | |
| | 0.4 | X | | WALL TEST |
| | 0.6 | 0.70 ± 0.03 | | |
| | 0.8 | 0.84 ± 0.05 | | |
| | 1.0 | 1.02 ± 0.03 | | |
| | 1.2 | 1.18 ± 0.02 | | |
| | 1.4 | 1.41 ± 0.05 | | |
| | 1.6 | 1.63 ± 0.05 | | |
| | 1.8 | 1.86 ± 0.03 | | |
| | 2.0 | 2.03 ± 0.04 | | |
|  | 0.2 | X | n=5 | |
| | 0.4 | 0.36 ± 0.02 | | GAP TEST |
| | 0.6 | 0.54 ± 0.02 | | |
| | 0.8 | 0.69 ± 0.07 | | |
| | 1.0 | 0.86 ± 0.02 | | |
| | 1.2 | 1.11 ± 0.12 | | |
| | 1.4 | 1.18 ± 0.04 | | |
| | 1.6 | 1.36 ± 0.04 | | |
| | 1.8 | 1.51 ± 0.04 | | |
| | 2.0 | 1.66 ± 0.04 | | |

Table 1: Measurements of wall thicknesses for various sizes, including mean +/- standard deviation.

| | wall test print | | | | | | | | |
|---------------|-----------------|------|------|------|------|------|------|------|--------|
| | 2 | 1.8 | 1.6 | 1.4 | 1.2 | 1 | 0.8 | 0.6 | mm |
| sample | | | | | | | | | |
| 1 | 12 | 92 | 76 | 51 | 22 | 10 | 95 | 75 | mm/100 |
| 2 | 2 | 85 | 63 | 39 | 16 | 1 | 83 | 68 | |
| 3 | 1 | 84 | 61 | 39 | 16 | 1 | 84 | 67 | |
| 4 | 0 | 84 | 60 | 40 | 17 | 0 | 81 | 67 | |
| 5 | 1 | 85 | 60 | 38 | 16 | 0 | 81 | 68 | |
| 6 | 2 | 85 | 59 | 38 | 17 | 0 | 90 | 69 | |
| 7 | 1 | 84 | 61 | 38 | 16 | 0 | 81 | 69 | |
| 8 | 2 | 84 | 61 | 39 | 16 | 0 | 81 | 70 | |
| 9 | 5 | 84 | 63 | 44 | 20 | 6 | 82 | 71 | |
| 10 | 8 | 92 | 70 | 47 | 19 | 3 | 84 | 75 | |
| | | | | | | | | | |
| mean | 2.03 | 1.86 | 1.63 | 1.41 | 1.18 | 1.02 | 1.84 | 1.70 | |
| stdev | 0.04 | 0.03 | 0.05 | 0.05 | 0.02 | 0.03 | 0.05 | 0.03 | |

Table 1: Measurements of gap thicknesses for various sizes, including mean +/- standard deviation. Note: 1.84 (0.8 measurement), should read 0.84 instead.

| | gap test print | | | | | | | | | | | |
|---------------|----------------|------|------|------|------|------|------|------|------|-----|-----|--------|
| | 2 | 1.8 | 1.6 | 1.4 | 1.2 | 1 | 0.8 | 0.6 | 0.4 | 0.2 | 0.1 | mm |
| sample | | | | | | | | | | | | |
| 1 | 71 | 49 | 42 | 14 | 28 | 90 | 81 | 52 | 38 | - | - | mm/100 |
| 2 | 67 | 48 | 37 | 16 | 19 | 86 | 68 | 55 | 39 | - | - | |
| 3 | 64 | 50 | 35 | 16 | 3 | 85 | 66 | 51 | 35 | - | - | |
| 4 | 62 | 51 | 35 | 19 | 2 | 84 | 64 | 57 | 33 | - | - | |
| 5 | 64 | 51 | 30 | 24 | 4 | 87 | 66 | 54 | 37 | - | - | |
| | | | | | | | | | | | | |
| mean | 1.66 | 1.50 | 1.36 | 1.18 | 1.11 | 0.86 | 0.69 | 0.54 | 0.36 | - | - | |
| stdev | 0.04 | 0.01 | 0.04 | 0.04 | 0.12 | 0.02 | 0.07 | 0.02 | 0.02 | - | - | |

Things to Avoid

- Do not print out pieces that could be easily be made by hand or with another machine e.g. laser cutter
- Very thin or small items will not print
- Long flat strips are hard to make as they tend to warp.
- Avoid unnecessary protrusions, especially on the base of the part.
- Avoid overhangs that will need support material. The Makerbots can handle up to 68 degrees from the vertical without support (normally).

Tips

- Cutting models up and adding locating holes can eliminate a lot, if not all support material.
- Make wall thickness as small as possible that the design will allow, but no less than 1mm.
- When designing your part think in what orientation it will be printed and the supports that it will need.
- Where supports are, think if you can include it in the model, then there will be less post processing removing supports, giving a better looking model.

Layers

- There is a weakness between layers

Holes

- Model holes bigger than they need to be to allow for shrinkage.
 - Holes smaller than 10mm diameter should be have a clearance of 0.4mm e.g. for an 8mm hole make it 8.4mm
 - For holes larger than 10mm, increase the clearance
 - 1mm clearance for 50mm holes i.e. 51mm
 - 2mm clearance for 130mm holes i.e. 132mm
 - Roughly scale the clearance for diameters between 10mm and 130mm e.g. 90mm would be modelled as 91.5mm
- Vertical holes are generally less geometrically accurate.

Parts that fit together

- Where a fit is required make smaller test pieces first
- Where two parts must fit together exactly, locating holes should be made each side and use rods to fit together. The rods could be made by cutting a small piece of the 3D printing plastic, which has a diameter of 1.75mm. To allow for shrinkage, model the holes as 2.2mm Alternatively, use welding rods, tin copper wire or enamel coated wire, but check availability of rod sizes first.
- Where two parts fit together with a lip, leave a 0.2mm gap on each side.
- When using snap fits make sure the print is in the correct orientation to avoid x,y plane weakness.
- Here's some free software (I think there's a plugin for Solidworks) to help slice up a model into several printable parts! <http://labs.cctech.co.in/3dprinttech/>

Solids

- If access to the interior of the model is not required make it solid.

- If it needs to be more solid we can stipulate an infill normally 5 – 50% to make it heavier and stronger.