

Instructions for using the CRR10:

Creality 3D - Our Trustworthy 3D printer

FABLAB BRIGHTON 2018



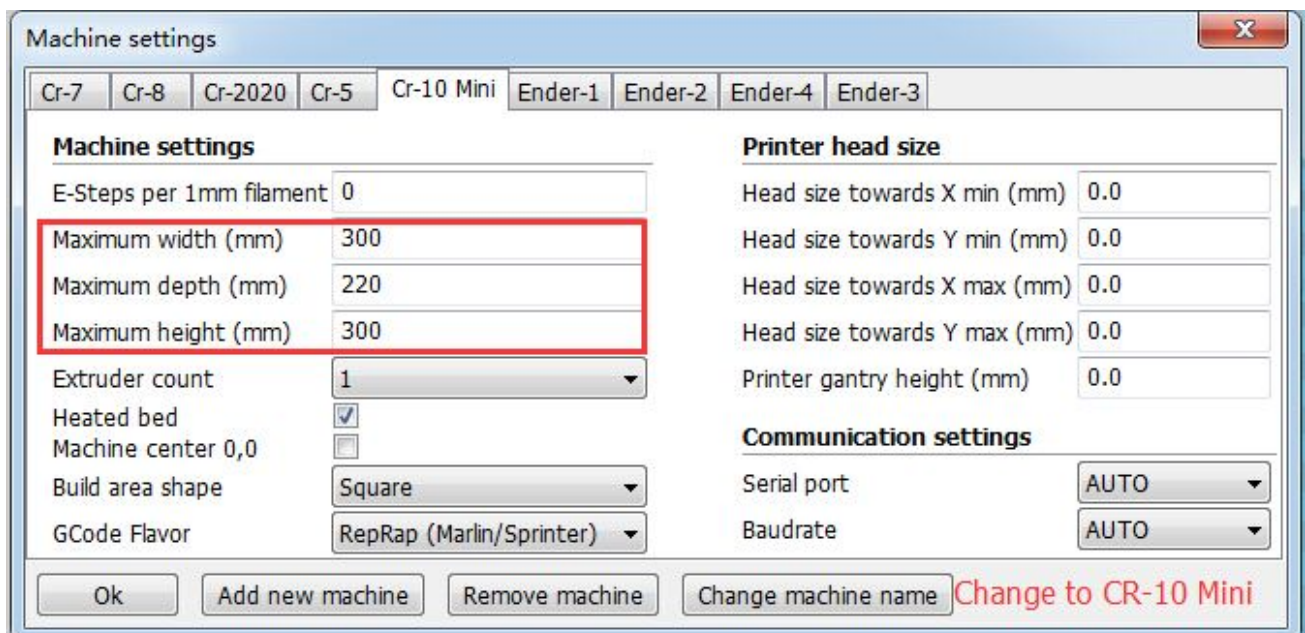
Installing Curamaker software

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:

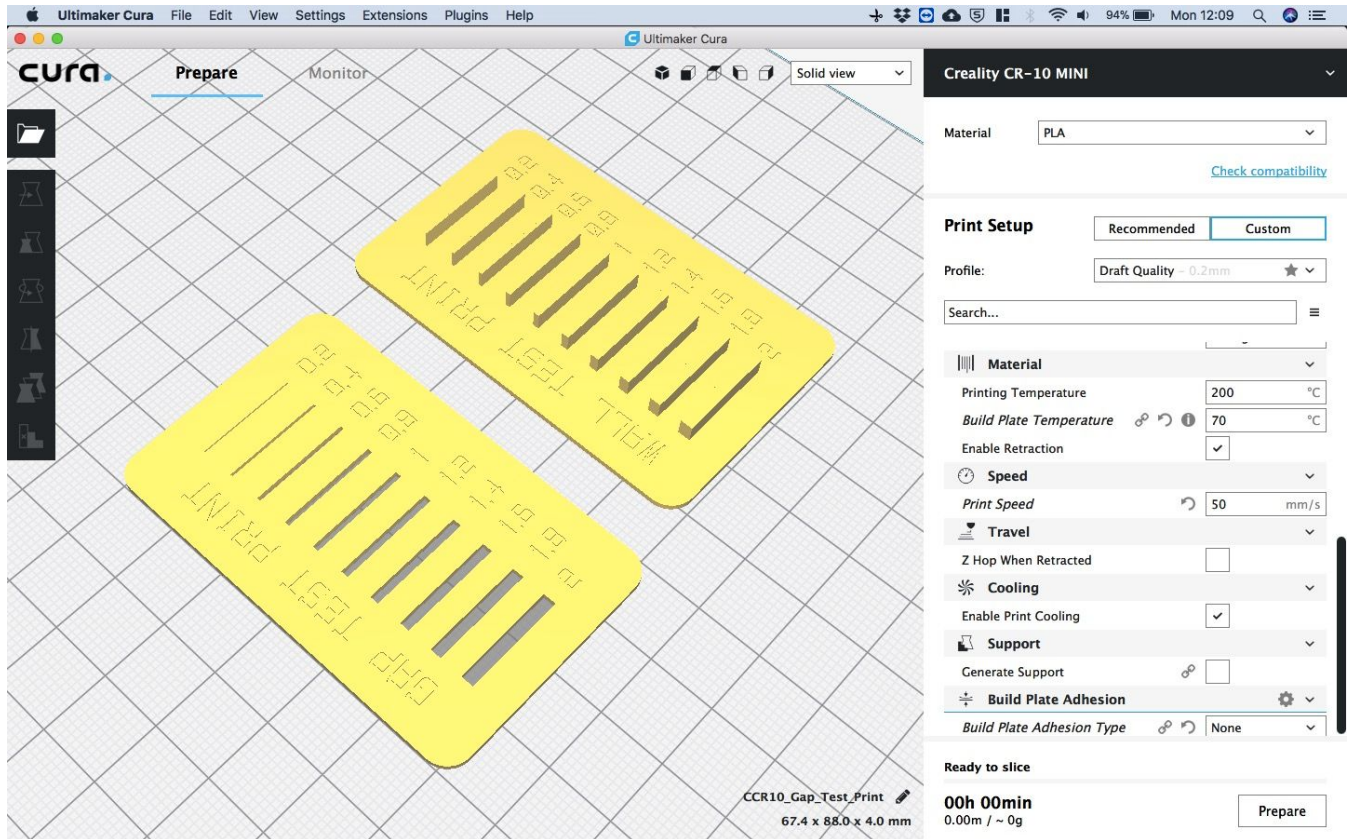


And then the main thing to change in the Custom print setup is the bed temperature...for this machine **THIS MUST BE 70 deg C**. The other big consideration is whether you want support material or not. Here are the basic settings we used to start with:

 Quality 	
Layer Height	 <input type="text" value="0.2"/> mm
 Shell 	
Wall Thickness	<input type="text" value="0.8"/> mm
Wall Line Count	<input type="text" value="2"/>
Top/Bottom Thickness	<input type="text" value="0.6"/> mm
Top Thickness	<input type="text" value="0.6"/> mm
Top Layers	<input type="text" value="3"/>
Bottom Thickness	<input type="text" value="0.6"/> mm
Bottom Layers	<input type="text" value="3"/>
Horizontal Expansion	<input type="text" value="0"/> mm
 Infill 	
Infill Density	<input type="text" value="20"/> %
Infill Pattern	<input type="text" value="Triangles"/> 
 Material 	
Printing Temperature	<input type="text" value="200"/> °C
Build Plate Temperature	   <input type="text" value="70"/> °C
Enable Retraction	<input checked="" type="checkbox"/>
 Speed 	
Print Speed	 <input type="text" value="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 	
Build Plate Adhesion Type	  <input type="text" value="None"/> 

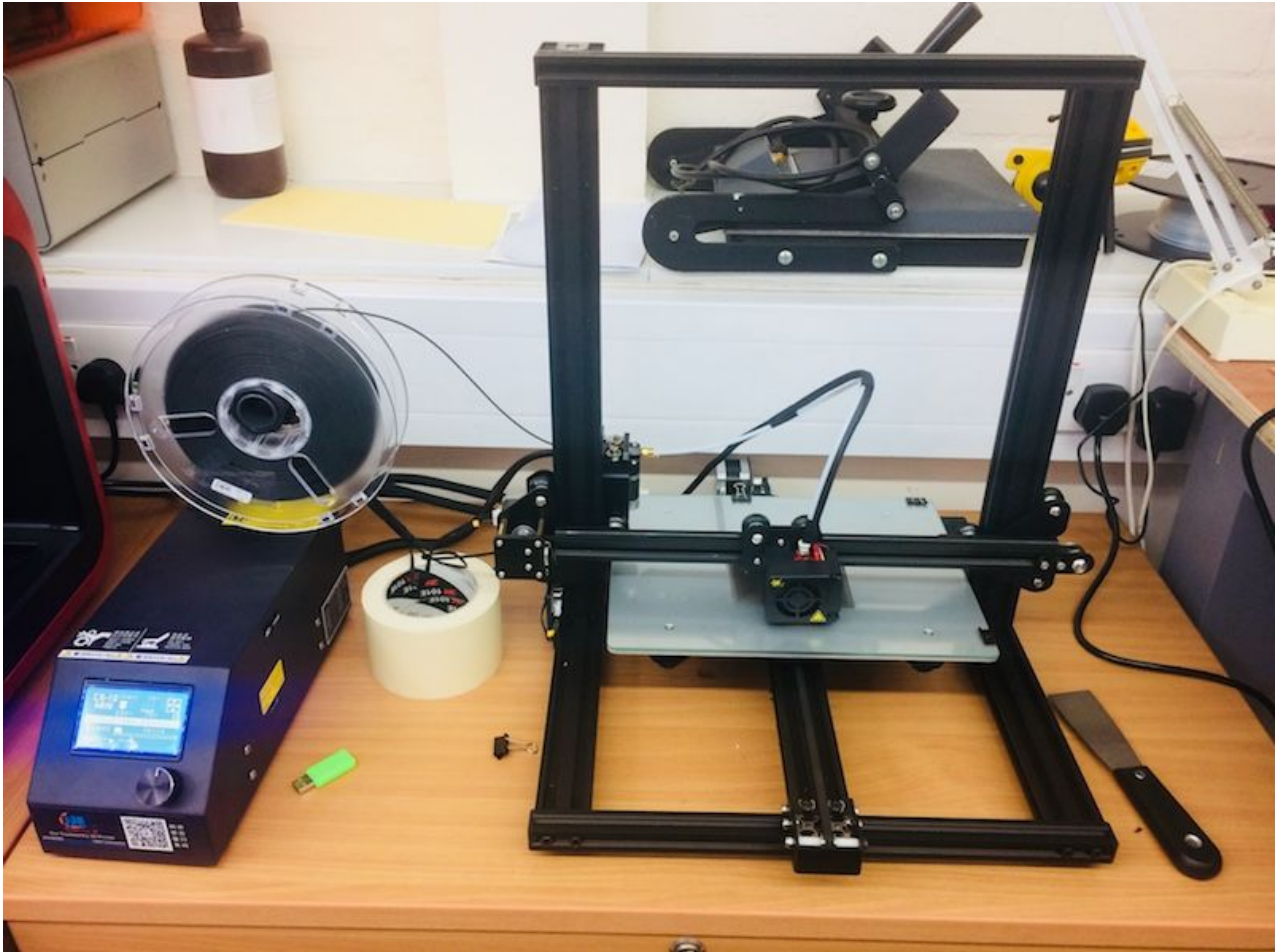
Loading your file into the software

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:

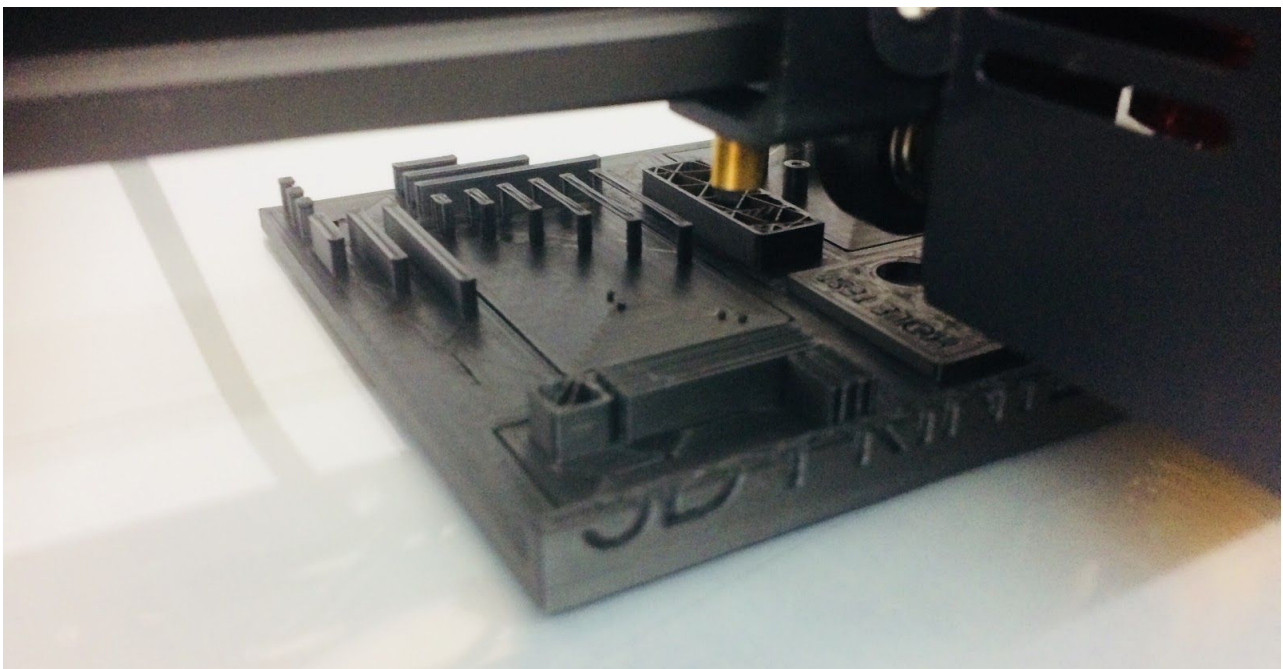


Once this is generated, Cura has a really nice feature, where if you plug in your USB stick, it will ask you if you want to save the G-Code directly to your USB device (button on bottom right again). Note that you can ONLY use the special FLUORESCENT GREEN USB stick. This has a small SD card embedded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

Search



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

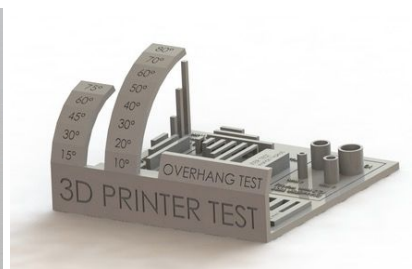
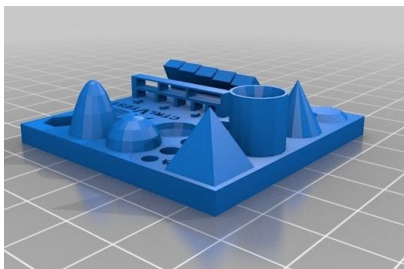
Unlisted

Printing calibration test models

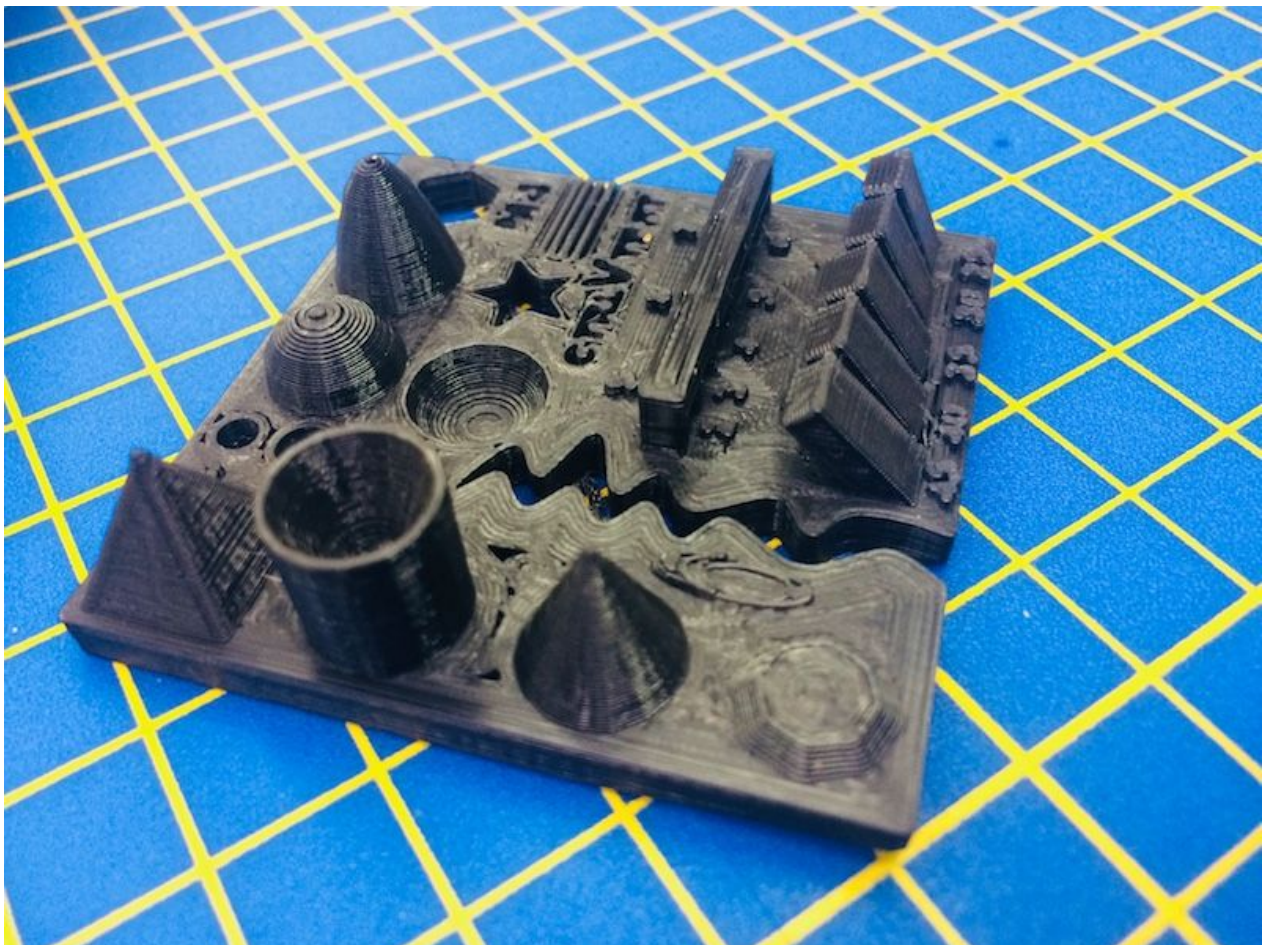
We then printed a series of test models to check the capabilities, design rules and limitations of this machine. And what a machine! We were very impressed by this little beauty. So easy to use, so impressive with reliability and repeatability (so far), and all for only about £300. Here are the three 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.

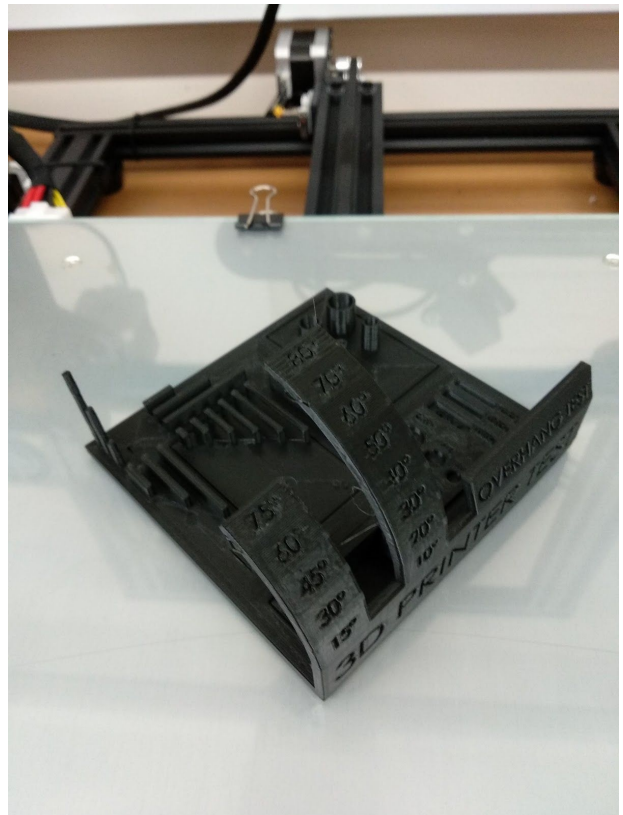
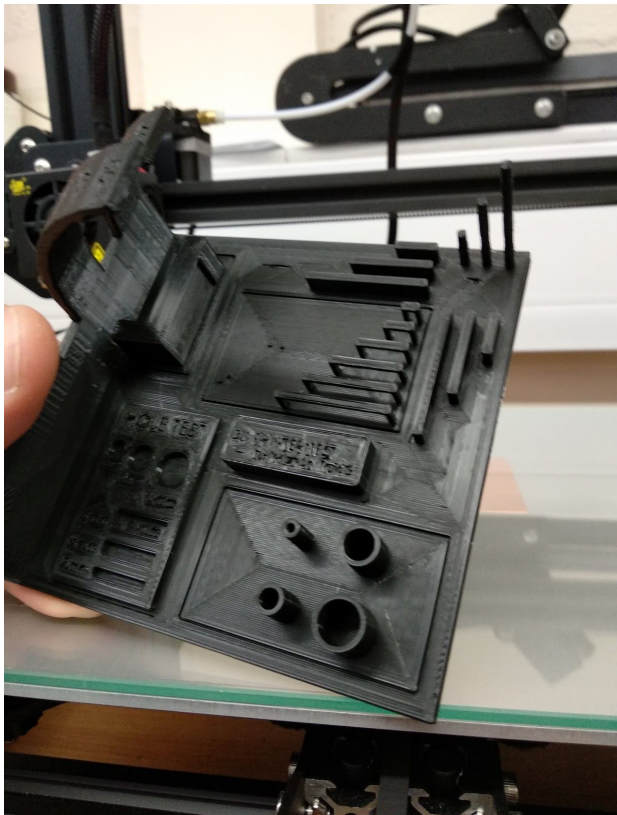
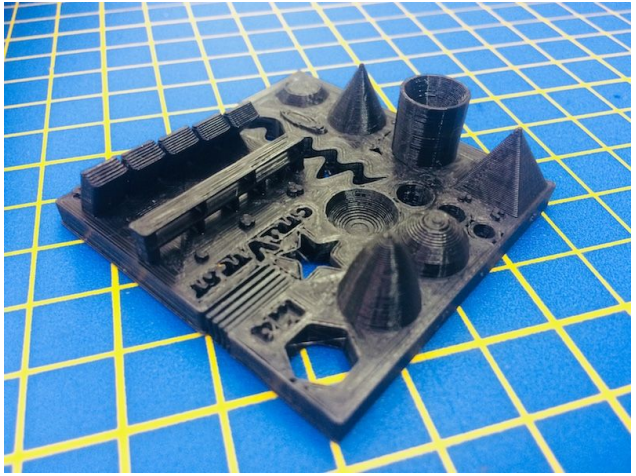
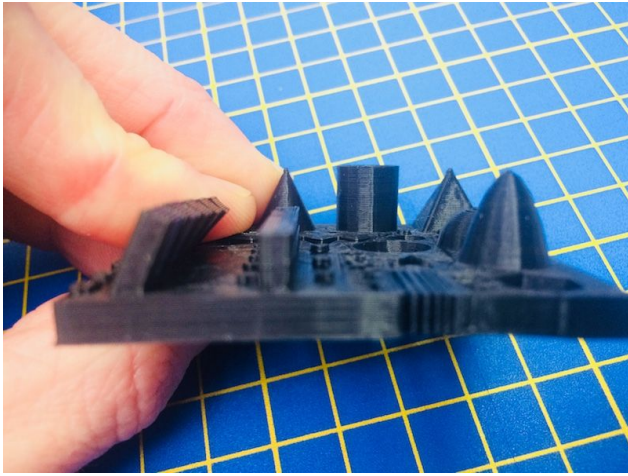
To summarise key outcomes:

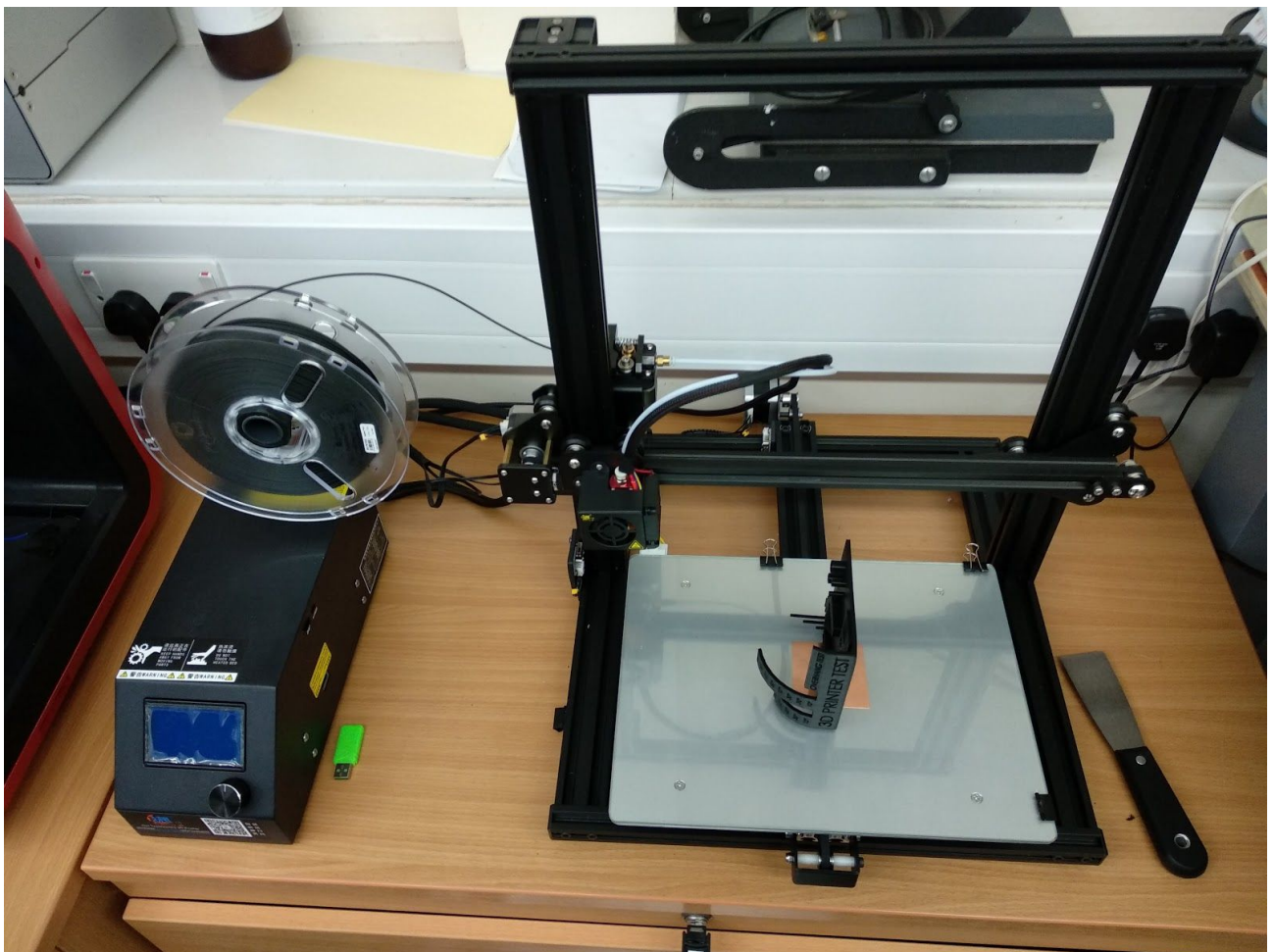
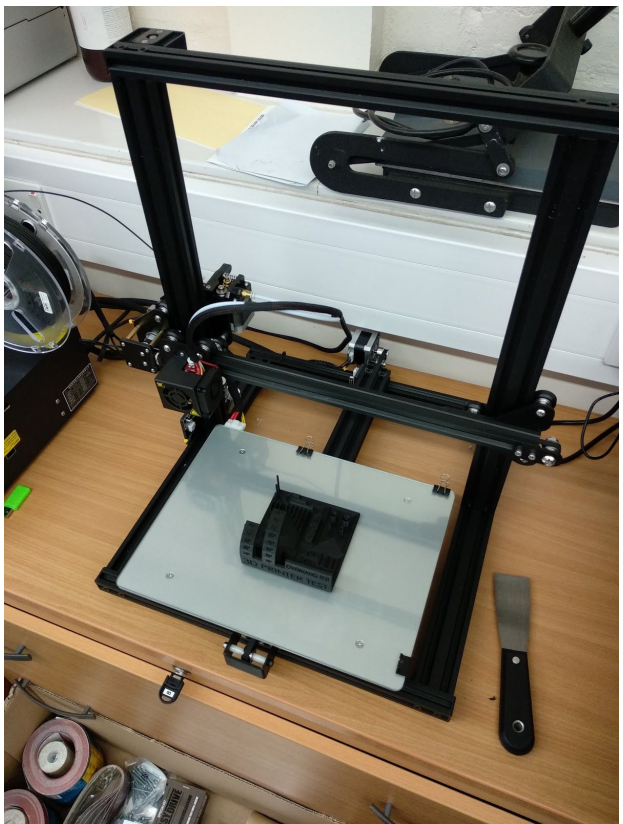
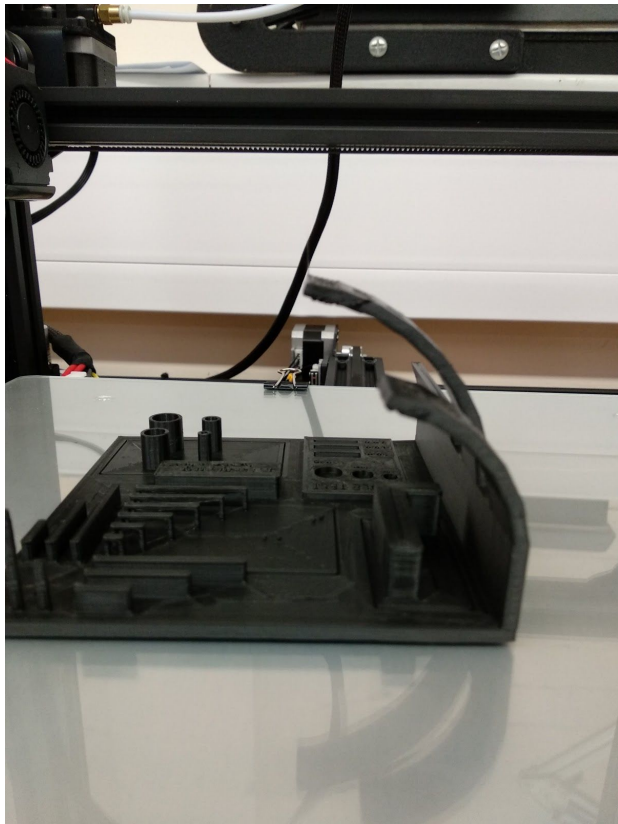
- We could only print standalone walls down to 0.6 mm wall thickness. 0.4 mm and 0.2 mm did not print at all. Note we didn't try 0.5 mm, but I suspect this would work.
- Walls tended to be larger than drawn (mean increase 0.03 mm)
- Gaps tended to be smaller than drawn (mean decrease 0.19 mm)
- Overhangs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

A black, rectangular cross-section of a part with several vertical ridges or walls of varying heights. The ridges are evenly spaced and have a consistent width.

SPECIFIED

MEASURED (mm)

0.2	X
0.4	X
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

n=10

WALL TEST

A black, rectangular cross-section of a part with several vertical gaps or slots of varying widths. The gaps are evenly spaced and have a consistent width.

0.2

X

0.4	0.36 ± 0.02
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

n=5

GAP TEST

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	-	-	

Makerbot Replicator 2X

We also have makerbots available to print work in the Advanced Engineering Centre. Here are the instructions and design guides which we hope are useful.

Designing Parts to be Printed on the MakerBot Replicator 2X

Saving

- Please save your file, as your name, what the part is called and the approximate size, colour and material i.e. Joe_bloggs_castle_50x50x40_blue_abs.stl
- Save your files as STL using (fine setting) in Solidworks, Rhino or equivalent and reload the STL file to check the curves are smooth enough.
- Save each item as a separate STL file

Size

- The maximum size is 220mm long x 140mm deep x 140mm high
- Bigger is not always better, only make it as big as it needs to be
- The extruders are typically 0.4mm wide. 0.4mm will give a single wall, 0.6 – 0.75 will give 2 walls that are merged. 1.25 is good for 3 merged walls. 1.5mm for 4 walls

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.

Checklist:

- Have you saved the files as separate STLs called your name, what the part is called and the approximate size, colour and material?
- Is the size no more than 220mm long x 140mm deep x 140mm high?
- Have you made allowances for holes to shrink?
- Would your design be more suitable to be made on a laser cutter?

How to get things printed at the University of Brighton

Step 1 - Prepare your CAD using the tips in the 'Designing Parts to be Printed on the MakerBots' section below

Step 2 - Save as an STL file/s

Step 3 - Complete the Checklist (on the left)

Step 4 - Bring the STL file/s on a USB stick and talk to Trevor or final year students or other students working on the printers. They will look at your CAD with you to check that it's ready and discuss how you would like it printed, such as the size and colour. If your CAD is not ready to print, they will make suggestions if possible

Step 5 - Your file/s will be transferred to the 3D printing computer from your USB stick and your details will be added to the 3D printing queue.

Step 6 - When your model is about to be printed it, the file/s will be transferred to one of the 3D printers using an SD card. Your model will then be printed.

Please note: each printer has a dedicated SD card, so you do not need to provide one. Your model may not be printed on the day it's added to the queue, depending on availability, so please allow extra time. You do not need to be present for Step 6, although you may want to watch your model being printed.

Step 7 - Once your model is printed it can be collected from outside room 101 in the Advanced Engineering Centre.