

**TOPIC: INTRODUCTION TO 3D PRINTING****At this workshop we will:-**

- A) Learn how to
- 1) Prepare the printer before the print job
  - 2) Find a simple pre-made design in Thingiverse.com to print
  - 3) Set up the parameters to "Slice" the model
  - 4) Transfer the resultant code to the printer
  - 5) Print the model..
- B) (If there is time), Learn how to
- Design a simple enclosure using Microsoft 3DBuilder;


---

**Topics:**

<b>A. PRE-WORKSHOP PREPARATION</b>	<b>Page 2</b>
<b>B. AT THE WORKSHOP-</b>	
<b>-Preparing the Printer for printing-Levelling the Bed.</b>	<b>Page 2</b>
<b>C. Testing the Bed Levelling procedure.</b>	<b>Page 3</b>
<b>D. Slicing the Bed Level Test Model</b>	<b>Page 4</b>
<b>E. Mounting filament</b>	<b>Page 5</b>
<b>F. Printing the Bed Level Test Model</b>	<b>Page 5</b>
<b>G. Selecting an existing Model to Print</b>	<b>Page 5</b>
<b>H. Cura Slicer Tools</b>	<b>Page 7</b>
<b>I. Designing your own Model</b>	<b>Page 8</b>

NOTE: In this document the word "Click" by itself means Left Mouse Click. "Right Click" will be specified only where necessary.

**A. PRE-WORKSHOP PREPARATION****Please do this at home in preparation for the Workshop**

1. Install [Ultimaker Cura](#) software onto a windows computer. Click [HERE](#) for supported versions of Windows, Apple & Linux.  
However, in this workshop we assume Windows 10 or 11.  
You **don't** need to create an Ultimaker or Makerbot account
  
2. Run the **Ultimaker Cura program** , and click at the top right of the screen on **Marketplace** to install the following plugins:-
  - a) Mesh tools
  - b) Parts for calibration
  - c) Settings Guide
  - d) Monoprice Select Mini V2If you can't see these, click on "load more"
  
3. Click at the top menu line on  
Settings> Printer> Add> -Monoprice Select Mini V2.
  
4. You can exit Cura by clicking on the top right hand side "x".
  
5. Check whether Microsoft 3DBuilder software is installed-  
click on the Windows Start icon   
In the search bar at the top type "3D builder" . If it appears, it's installed
  
6. If 3D Builder is not installed, find the Microsoft Store icon on your PC, click on it, then in the App Search Bar at the top, type In  
3D Builder <enter>. It should be found and shown as a large icon. Click on the icon and either you will see "open" or "get" . OPEN means it's already installed. If not, click on GET to install it.

**B. AT THE WORKSHOP**

Using your Windows Computer:

**Preparing the Printer for printing-Levelling the Bed.**

Levelling is one of the most important steps. If the bed is not perfectly level, random problems will manifest themselves. For example, the model not sticking on the bed, or moving during printing causing "Spaghetti" because the filament is not sticking to itself and/or printing in thin air .

**Usually levelling is only required infrequently, mainly when the hardware is changed or adjusted (hotend, bed top material, or printer moved to another location).**

Each 3D printer has one of the following methods to level the bed

- Auto-Level, I manually initiated as required
- Levelling by printer automatically before each print
- Auto-level during printing
- Firmware Assisted Manual Levelling
- Manual Levelling

As the first four are printer dependent, we will leave that to be discovered by participants reading their manual depending on which printer they will use.

However, Manual levelling is pretty much the same for most basic printers.

The steps are:

1. Clean the surface of the printer bed. Some print surfaces require just a damp or wet cloth, others with IPA (Isopropyl Alcohol {usually about 70% pure})
2. Heat up the printer bed to the working temperature of the material you intend to use. PLA=60°, PETG=70° or 80°, ABS=100°.
3. Heat up the hotend to the working temperature of the material you intend to use. PLA=200°, PETG=230°, ABS=240°.
4. Retract any filament from the hotend and make sure the hotend has no filament drooping down. A brass brush or similar can be used to clean the hotend tip.
5. Next step is to "Home" the bed using the printer's control panel. This places the tip of the hotend at the X=0,Y=0, and Z=0 axes . (X is left-right, Y is forward/back and Z is height from the bed surface).
6. Test the position of the hotend above the bed by sliding a piece of A4 paper between the hotend and the bed. A good height is when the paper slides in and out with a light resistance.
7. If that's not the case, you should use the bed adjustment screws to lower or raise the bed at that corner. Try a quarter turn and test again. When successful, do that for each of the other three corners by moving the hotend either manually or via the printer's control panel. Then do the same at the centre of the bed.
8. Finally home the hotend (using the printer's control panel) and test that corner again. Adjust using the bed adjustment screws as necessary.
9. When satisfactorily complete, turn off the heating of the hot end and bed.

Course participants will not actually level the bed as there will only be one printer, but the process will be demonstrated.

### C. Testing the Bed Levelling procedure.

1. Run the **Ultimaker Cura** program.
2. In Cura, the 4th line should show (accross)
3. **Monoprice Select Mini E3D, Generic PLA,Normal 0.175mm,22% ,off,off**
4. If it's not, click on the field that doesn't conform to the above and select the correct parameter.
5. Next, from the top row menu click on Extensions>Parts for Calibration>Add a Bed Level Calibration. A calibration model will appear on the bed.

6. Using your mouse, right click & keep holding the right mouse button, and move the mouse upwards until you can see the bottom of the model. You will notice two different blue colours. The **light blue** means that the model is flat on the bed. The **dark blue** means the model is a bit below the bed. If the bottom is **red**, it means the model is not touching the bed and is above it.
7. To place the model perfectly on the bed, click on the model and then click on the blue up arrow, drag the arrow up (whilst holding the left button of the mouse), then let go. The model should drop to the bed automatically.
8. If not, click on "Preferences" from the top menu bar, then click on "Configure Cura>General>Defaults". finally close the preferences window by clicking once on the "X" on the top RHS of the Preferences pop-up window. (DO NOT close CURA!)
9. Then try to re-do the model bed placing procedure above in step 1.6

#### D. Slicing the Bed Level Test Model

This process takes the three dimensional model and creates a set of instructions that lays down two dimensional slices the height of about 0.2 mm , successively growing one on top of the other to create the 3D object.

The rightmost tab is the Print Setting Tab and has many options. At this point we will just use the defaults for PLA.

1. Click on the "Slice" button on the bottom RHS of the screen..
2. When finished , click on the "Preview" button and move the slider on the RHS down and up to simulate the printing. You are looking for obvious errors and unintended features.
3. One of these is a "Brim" which is a vertically thin but horizontally wide perimeter around the model designed to enhance adhesion to the print bed if there is insufficient width in the model to provide good adhesion.
4. In this model, and given good bed levelling, we don't need it. So we need to turn it off and re-slice.
5. To turn it off, scroll down the Print Settings window on the RHS of the screen, down to "Build Plate Adhesion" Click on the "v" on the RHS of the word "Brim", and select "None". The re-slice by clicking again on the "Slice" Button.

At this point we need to transfer the resulting sliced file (the "gcode" file) to the printer. There are usually 3 ways:-

- I. Save to an SD card on the PC, and transfer the SD to a card slot on the 3D Printer.
- II. Plug the PC into the printer using a USB cable and the PC becomes the printer controller for the whole duration of the print using Cura as the engine.
- III. Send the file to the printer over wifi

For simplicity, we will use the first method above.

- Insert a printer compatible SD card into your PC and click the “Save to Disk” Button on the bottom RHS of the screen, and name the file “Bed Level”.
- Remove the SD card, and insert it into the 3D printer.

#### **E. Mounting filament**

Next we need to feed the filament into the printer.

The Steps are:

1. Use the printer’s control panel to move the print head to about 60mm above the bed.
2. Next, heat up the hotend to 200° C from the printer’s control panel.
3. When the hotend reaches 200°, use the control panel to “**extrude**: the filament till it leaves the hotend in a steady stream for about 60mm.

#### **F. Printing the Bed Level Test Model**

*Some filament and bed surface combinations may require additional surface coating or preparation. In this course we will assume we don’t need to.*

Using the printer’s control panel, select “Print” and select “Bed Level” as the file to be printed.

Now wait for it to finish. It should be quick, as the bed level model is designed to take minimal time and minimal material. Estimate is about 4 to 6 minutes.

- When the print is finished examine the printed model.
- Did it print as expected?
- Did it all stick to the bed?
- Are all the squares equal in size and height?
- Do the top and bottom layers look smooth?

If all is good, we can print an actual usable model.

#### **G. Selecting an existing Model to Print**

There are many sources of existing crowdsourced models, mostly free, but some expecting some payment.

Two of the most popular web sites are:

[Thingiverse.com](https://www.thingiverse.com)

&

[Printables.com](https://www.printables.com)



A more complete list is [HERE](#)

Before you try to design something, have a look whether others have already designed the exact thing, or even something close you can modify to your needs.

Today we’ll pick a simple box

<https://www.thingiverse.com/thing:2087914>

Click on “Download all files” and remember where the zip file is saved (usually to “Downloads” on your PC.

1. Open windows explorer (click on the  icon) on your PC.
2. Double click on the “Simple Box with lid 2087914.zip” file and then click on “Extract All” on the 2nd line from the top of the screen.
3. Remember where the files are extracted to (usually in the same directory as the zip file).
4. In Cura, click on the “folder” icon  at the top LHS of the screen. A file selection window will open, probably in the same folder you saved the **box.stl** and the **lid.stl** files.
5. Double click on the “**box.stl**” and see it appear in the Cura workspace
6. Do the same for the **lid.stl**.
7. Now the box and lid are both in Cura, but they may be overlapping each other. In that case, to separate them, click on the top left menu of Cura **Edit>Arrange all Models**
8. Check that the models rest flat on the bed (remember light blue from the bottom is good?)
9. Otherwise refer back to Page 4,C(7) to fix it.

So then all we have to do is click on the “Slice” button and follow the same directions as on Page 4, Heading D & F.

## H. Cura Slicer Tools

Cura is a very powerful software. Program. It can be configured to only show “basic” controls, or a range between Advanced, Expert or All. Or even a customised subset.

To change this view, click on the three horizontal bars in the print settings window (to the right of the search bar). The available parameter settings are almost endless. However, each one has both an effect on many others, as well as unintended consequences.

If you want to understand what each setting does and the consequences, hover your mouse pointer over the setting ,and a very comprehensive settings help system will appear.

(you did install the “Settings Guide” plugin as per Page 1, A(2)(c) didn't you? )

### Other Common Controls (simple editing features)

Other more common basic controls are on the icons on the left hand side. If you hover your mouse over them they will give (less comprehensive) tool tips. They will only be available (i.e. Not greyed out) if you select a model by clicking on it..



Move

The first is the **MOVE** icon. You can drag the model around the workspace



Scale

Next is the **SCALE** icon. You can make the model bigger or smaller, either keeping aspect ratio or just individual X,Y or Z dimensions



Rotate

**ROTATE** allows you to rotate the model in any of the X,Y or Z directions, and snap To with 90 degree steps, or individual 1° steps.



Mirror

**MIRROR** allows you to mirror the model in any of the X,Y,or Z directions.



Per Model Settings

**PER MODEL** allows different models on the same workspace to have some different printing parameters.



Support Blocker

**SUPPORT BLOCKER** allows individual models in the same workspace to be excluded from support structures. (Supports allow overhangs to be printed).



Measure

**MEASURE** allows you to measure sizes that are not shown on the outside of the model. For example, the inside dimensions of a box.




YOU CAN NOW EXIT CURA BY CLICKING ON THE “X” ON THE TOP RHS OF THE SCREEN.

**I. Designing your own Model.**

Let's make a simple container with a lid using the free and inbuilt Microsoft 3D Builder software. Yes, you'll need Windows 8,10 or 11!

1. Click on the Windows Start icon  and type "3D Builder". The 3D Builder app will appear, so click on it to start the program
2. Then Click on "**New Scene**"
3. At the top menu line Click on **Insert>Cylinder**
4. A cylinder will appear, as well as some editing tools as below



5.  So first we'll resize this by clicking on the Scale tool . It will show the current dimensions in mm, which are X=40 Y=40 and Z=40.
7. Let's change the model size to 50,50,50 by clicking on each of the numbers, then typing the new number of 50 in each dimension and clicking on accept each time.
8. We could have also made it bigger by clicking the little padlock icon  on the RHS of the tool so it changes to a locked padlock. This locks the aspect ratio so we can change all of the dimensions in one go.
9. However, by making the Z dimension 50 mm, the cylinder now protrudes downwards below the imaginary bed. So let's put it back flush with the bed.
10. So click on **Object>Settle>**√ (click on the "tick")
11. Next we'll make this cylinder hollow, so click **Edit>Hollow**
12. A dialog will appear suggesting the outer wall minimum thickness to be 1mm. Change that to 2mm by dragging the slider to the right and then click on "tick" √
13. It doesn't look like anything has changed because the outside walls are hiding the hollow inside. So let's cut off the top but keep it for a lid..
14. Click on **Edit>Split**. Then click on "**Keep Both**".
15. A box will open at the bottom which shows a number and mm. Type in **-15**, (**that's minus 15**) and then click on the "tick" with the word "split" next to it.




16. There should be a panel on the RHS of the screen that allows, amongst other things, to view which model is selected. If you can't see the panel, click on the "<" sign at the top of the panel to make it wider.

17. Now when you click on each of the two bits of the cylinder, one should be greyed out indicating it is not selected.

18. So select the top part, which will be the lid.


19. Click on the "Scale" tool  at the bottom middle of the screen


20. What we are going to do is to make this top bit the lid that will fit around the outside of the main cylinder. We specified 2mm walls, so two sides of those are 4, and we'll add another 1mm for clearance. So we need to scale the X & the Y to 60.. We don't want to let the Z grow in height too because it will take longer to print for very little

space gain. So we'll leave the padlock unlocked  to allow individual dimension adjustments.

21. Click on each of the X & Y dimensions and change them to **60** each. See how the lid grew?

22. So now we will move the lid to the right to clear the main cylinder, rotate it so the flat side is on the bed, and then drop it to bed.

23. Click on the Move tool  and drag the lid to the right past the cylinder.

24. Next click the rotate tool , click on "roll" and type **180 <enter>**. The lid will flip 180 degrees.

25. Lastly we drop the lid to the bed by clicking on **Object>Settle>** ✓.

**26. We are finished with the design, let's save it.**

27. Click on the "save" icon  at the top right of the screen

28. Finally, in order to print we need to export the model to a file that we can import into Cura.

29. Click on the three horizontal bars  at the top left of the screen and click on "Save As" naming it "Round Container"

Normally we would then load the model into Cura, Slice it, and then send the model to the 3D Printer to be printed.

We won't do that today because it will take about two and a half hours to print.

{The End}

**ANY QUESTIONS?**