

### ***Introduction:***

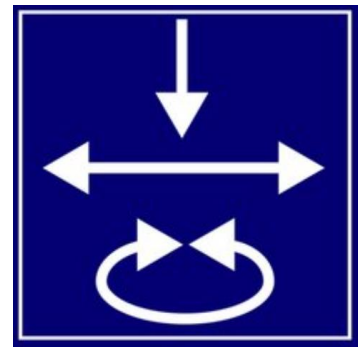
Hello and welcome to the guide on how to operate “Polar Polly” – Large Format 3D Printer. This printer is not as straight forward to use as the other consumer and industrial machine that are in the makerlab, therefore we will start with going over the background of the machine, its pros and cons, what software is needed and what considerations to make before printing on this 3D printer.

### ***History:***

This machine was kindly donated to us from DTU. When it arrived, it was not working, and it was clear that it needed to be upgraded to be a reliable piece of machinery. The mechanics of the printer are solid and of high quality, but the electronics were horribly outdated. It was upgraded to a state-of-the-art DUET3D mainboard and RaspberryPi combination.

### ***“Polar Printer”:***

The kinematics of this machine is called “Polar printer” it is very uncommon and very few machines like this exist on the market. It works like so; the Y-axis is a rotary table that can rotate an infinite number of degrees in both positive and negative. The X-axis is a radius arm that moves in and out toward or away from the center of the rotary table. The Z-axis lifts or lowers the radius arm.



### ***Print area:***

The Polar Printer kinematic has the advantage of enabling a machine to have a large print area together with the whole printer being quite compact. There are of course drawbacks with this kinematic, one drawback is that the center of the build plate is unusable due to limitations in the rotary speed of the table. Because of this you must consider the print area being a doughnut shape with a ~100mm hole in the center. This is not possible to set up in the slicer and it is therefore up to the user to never place models in the center of the build plate.

### ***Print area:***

Diameter: 750mm

Height: 1250mm

## Software:

For slicing your files, you will use CURA, the same software as for using the Ultimakers. Attached there will be two custom profiles that will have to be added to CURA and used for slicing files.

The printer is an FDM printer just like the Ultimaker and Markforged machines elsewhere in the makerlab. Meaning that it melts a thermoplastic and deposits it in space according to the .gcode file that is input.

To produce .gcode for the machine you will use CURA with a specific profile that is tuned to work with the machine. We will cover this in detail later in the guide.

To control the machine and send your .gcode files to it you must connect to the printers User Interface over Wi-fi. The printer is using a DUET3D mainboard as the muscle and a RaspberryPi as its brain and it is this that is accessed directly over wi-fi in your browser.

### Step 1: Connect to the DUET UI over wi-fi

Connect to the following wi-fi network:

Name: TP-Link\_627D

Password: andh@kea.dk

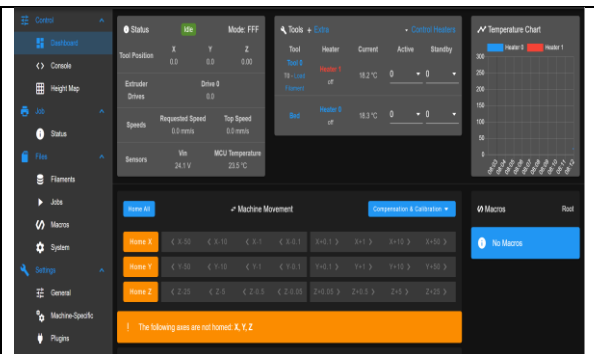
#### Step 1.1:

In your preferred browser (Chrome, firefox, edge is recommended) type in the following ip-adress:

192.168.1.101

This will take you to the DUET UI.

(If the page doesn't load make sure that both the RaspberryPi and the DUET3D mainboard are powered on and that you are connected to the correct wi-fi network.)



## Step 2: Generate .gcode

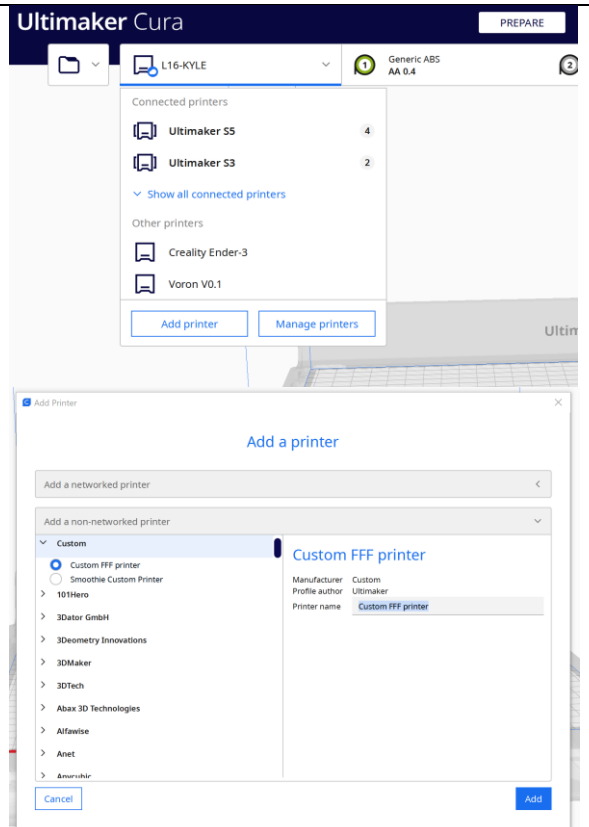
Setting up CURA and generating .gcode.

### Step 2.1: Add the printer to CURA.

In CURA press “add printer” followed by opening the drop-down menu called “Add non-networked printer”. Then choose “Custom” and then click “Custom FFF printer”.

Change the name to something of your liking and press “Add”.

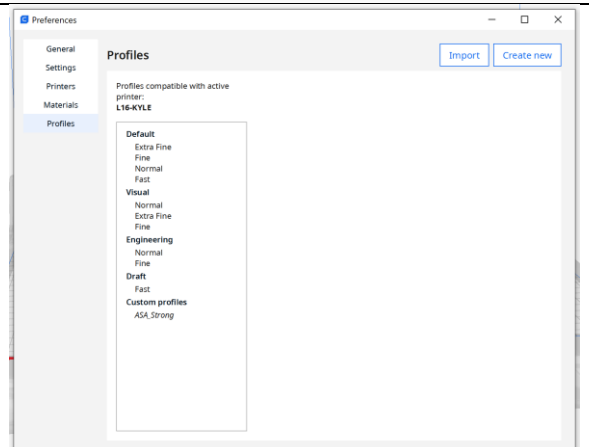
You will now have to copy the settings from enclosure 1 into CURA.



### Step 2.2: Import profiles

In CURA go to preferences -> Configure CURA -> Profiles -> Import.

This allows you to add the enclosed profiles that will work with the machine.



### Step 3: Start printing

Upload your .gcode file to the DUET interface using the “upload .gcode” button in the “jobs” tab.

#### Step 3.1: Start your print

Click on the file you just uploaded and press “yes” to print now.

#### Step 3.2: First layer / baby stepping

Carefully observe the first layer of the print. Being that this printer doesn't have a heated bed, adhesion of the print to the print bed is a challenge. Use glue stick or painters' tape to help the print stick. It is possible to fine-tune the distance from the build plate using “baby stepping” on the “status” tab.

The image shows two parts of the DUET interface. The top part is the 'G-Code Directory' which lists several files with their sizes, last modified dates, object heights, layer heights, filament usage, print times, simulated times, and the software used to generate them (Cura\_SteamE 5.2.1).

Filename	Size	Last modified	Object Height	Layer Height	Filament Usage	Print Time	Simulated Time	Generated by
G27WheelAdapter.gcode	7.9 MB	1/15/2023, 8:10:48 PM	30.00 mm	0.20 mm	11034.2 mm	1h 52m 44s	n/a	Cura_SteamE 5.2.1
BatteryTray_25p.gcode	1.5 MB	1/15/2023, 5:54:20 PM	31.00 mm	0.20 mm	4968.8 mm	1h 5m 38s	n/a	Cura_SteamE 5.2.1
Benchy.gcode	2.8 MB	1/15/2023, 4:05:52 PM	48.00 mm	0.20 mm	4020.9 mm	40m 40s	n/a	Cura_SteamE 5.2.1
18650x3Box.gcode	19.1 MB	1/15/2023, 12:13:54 PM	68.80 mm	0.20 mm	18020.3 mm	2h 18m 11s	n/a	Cura_SteamE 5.2.1
grid fin.gcode	2.7 MB	1/15/2023, 11:43:04 AM	98.00 mm	0.20 mm	5658.0 mm	1h 14m 34s	n/a	Cura_SteamE 5.2.1
CubeOcto.gcode	6.0 MB	1/14/2023, 6:50:44 PM	56.60 mm	0.20 mm	10192.8 mm	1h 32m 28s	n/a	Cura_SteamE 5.2.1
calcat.gcode.part	962.9 KB	1/14/2023, 6:28:00 PM	n/a	n/a	n/a	n/a	n/a	n/a

The bottom part of the image shows the 'Z Babystepping' control panel. It displays the current offset as 0.00 mm and provides buttons to adjust the offset by -0.05 mm and +0.05 mm.

# Enclosure 1

Machine Settings

### Polar Printer 1.2mm Nozzle

Printer	Extruder 1		
<b>Printer Settings</b>	<b>Printhead Settings</b>		
X (Width)	750.0 mm	X min	-20 mm
Y (Depth)	750.0 mm	Y min	-10 mm
Z (Height)	1250.0 mm	X max	10 mm
Build plate shape	Elliptic	Y max	10 mm
Origin at center	<input checked="" type="checkbox"/>	Gantry Height	1250.0 mm
Heated bed	<input type="checkbox"/>	Number of Extruders	1
Heated build volume	<input type="checkbox"/>	Apply Extruder offsets to GCode	<input type="checkbox"/>
G-code flavor	RepRap		
<b>Start G-code</b>		<b>End G-code</b>	
<pre>G28 ;Home all G29 ;Mesh bed level G29 S1 ;Load heightmap</pre>		<pre>M104 S0 M106 S0 P1 G90 G1 X375 F1000 M84</pre>	

Machine Settings

### Polar Printer 1.2mm Nozzle

Printer	Extruder 1
<b>Nozzle Settings</b>	
Nozzle size	1.2 mm
Compatible material diameter	1.75 mm
Nozzle offset X	0.0 mm
Nozzle offset Y	0.0 mm
Cooling Fan Number	1
<b>Extruder Start G-code</b>	<b>Extruder End G-code</b>