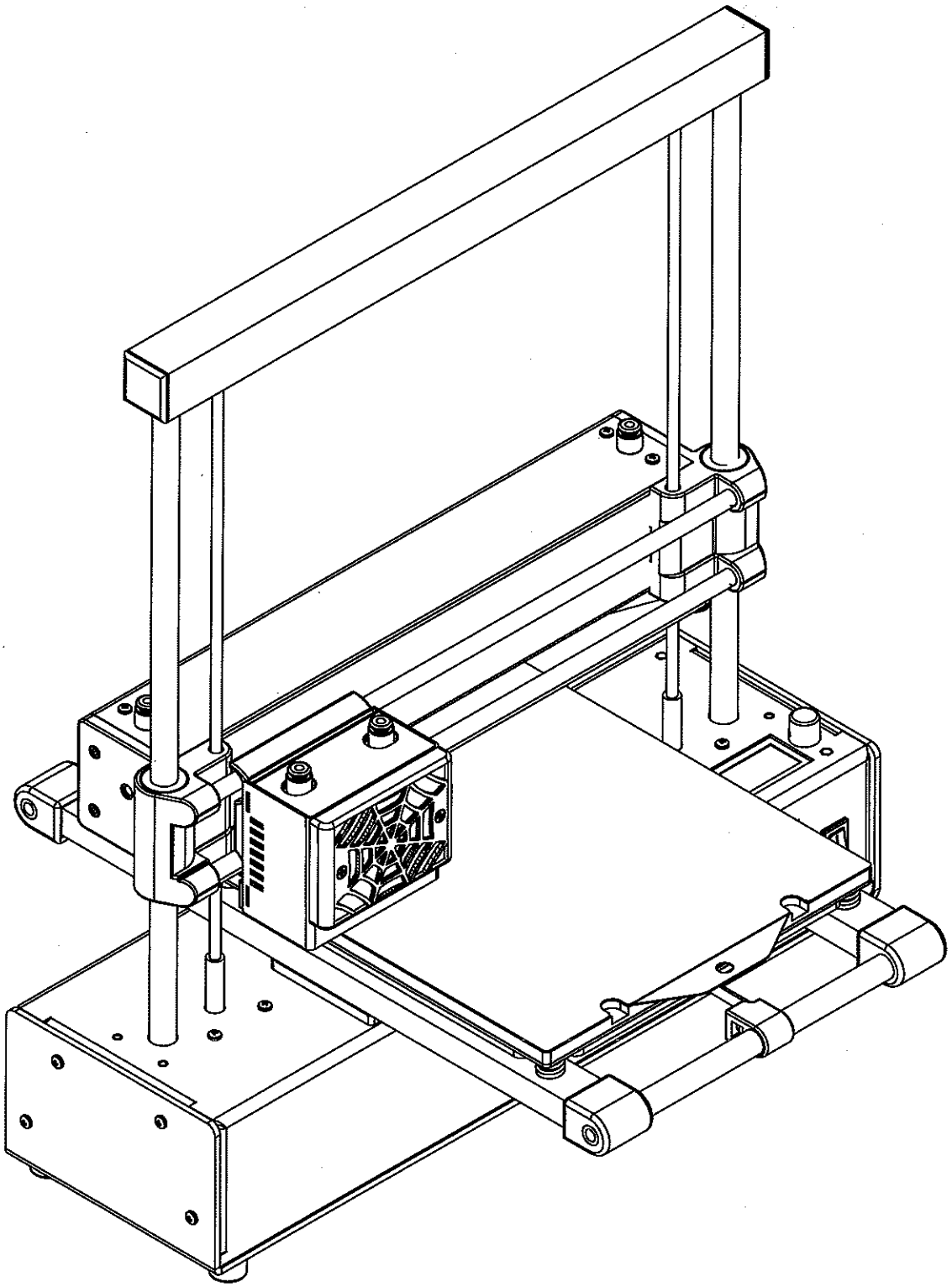
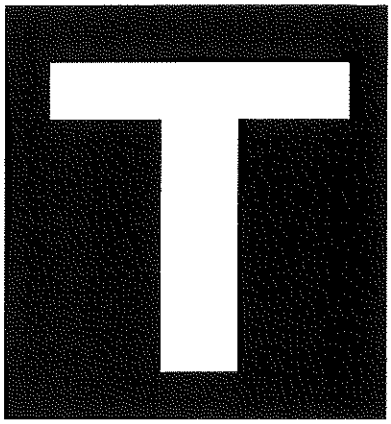
MAKE IT

Pro 2 3D Printer Manual



DRAFT

TABLE OF CONTENTS



SAFE OPERATION

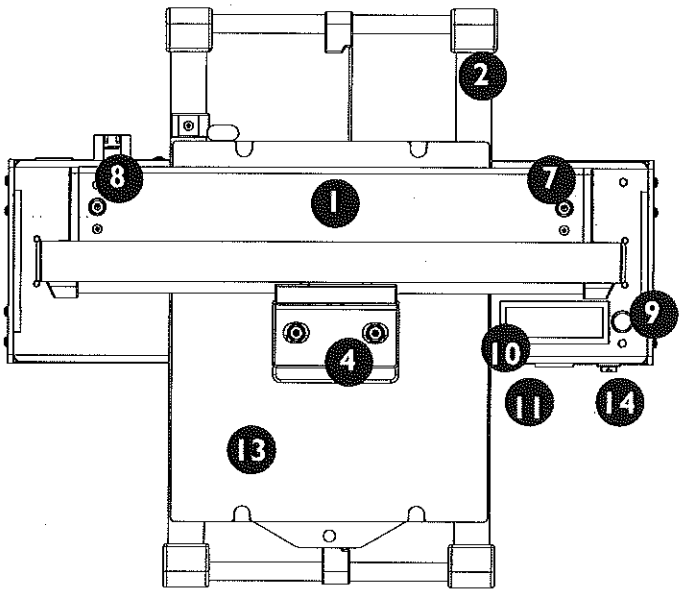
The MAKEiT printer uses the Fused Filament Fabrication (FFF) method of production (also knowns as FDM) which requires heating of the print material to near its melting point. The extruder module of the MAKEiT printer can reach temperatures up to 275o C/527oF, and the build platform up to 120oC/248oF. Working with such heated equipment, cautions are always needed during operation.

1. To avoid personal injury and part deformation, wait until the extruder module and build platform have cooled considerably before attempting to remove the printed part. Do not touch the extruder module, build platform, or part at any time during the print or immediately after the printer has finished printing.
2. In cases where it is necessary to operate the printer while it is hot, such as exchanging printer nozzles, we have included a pair of gloves and proper tools for your safety.
3. Any exposure to water or other fluids is likely to cause damage to the printer.
4. The operational ambient temperature range is between 60°F and 90°F [15°C and 32°C], humidity between 20% and 50%. Attempting prints outside of these ranges will result in reduced final quality.
5. Place your MAKEiT printer on a hard and sturdy surface, never leave it on fabric surface or carpet.
6. Operate in a well vented area to ensure adequate airflow to avoid printer overheating, and ABS fume.

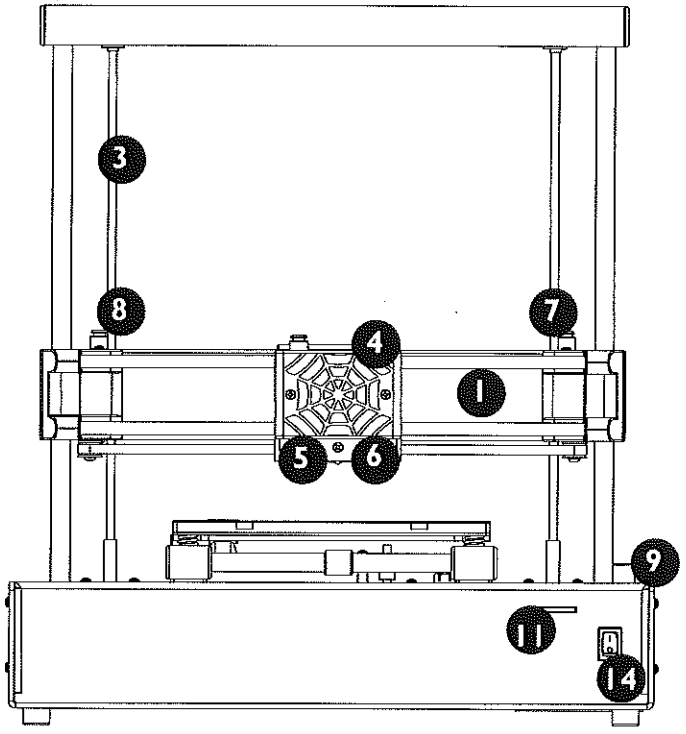
ABOUT THE PRINTER

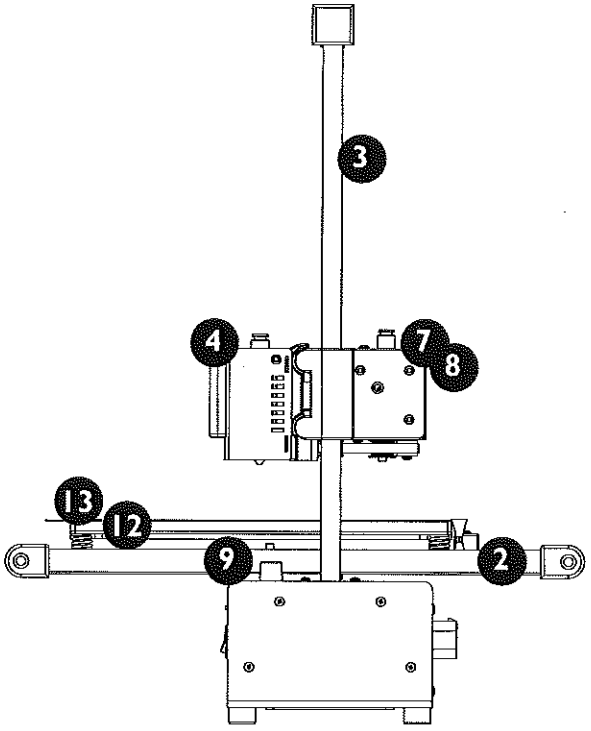
2

2.1 PRINTER OVERVIEW

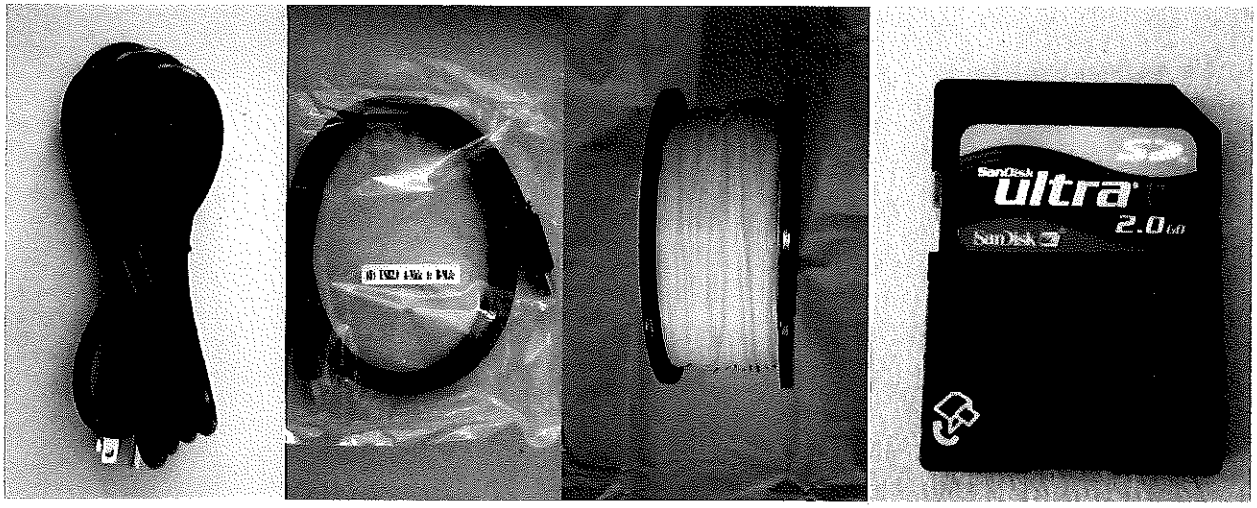


1. X-Bar
2. Y-Axis
3. Z-Axis
4. Printhead Module
5. Nozzle 1
6. Nozzle 2
7. Feeder 1
8. Feeder 2
9. Encoder Switch
10. LCD Display
11. SD Card
12. Heated Bed
13. Printer Plate
14. Power Switch





2.2 ACCESSORIES



Power Cable USB Cable Starter Filament SD/USB Card

SD Card Contents: MAKEiT custom firmware .ini files, printable STL files with GCode: build plate handle, filter cover (these can be personalized by user)

2.3 Tool Kit

Tool Kit Contents:
-one metal brush for cleaning
-one 7mm hex nut driver
-one 2.5mm hex driver
-one putty knife
-one lubricant
-one pair of gloves

2.4 PRINTER DATA SHEET

PRINTING

Print technology:	FFF, Filament Fused Fabrication (also known as FDM)
Build volume:	20 L x 24 W x 20 H cm [7.87 x 9.45 x 7.87 in]
Print quality / layer height:	>50 microns [0.05mm or 0.002in]
Positioning precision:	X axis: 10 microns [0.0003937 in, 0.01mm]
	Y axis: 10 microns [0.0003937 in, 0.01mm]
	Z axis: 0.5 microns [0.000019685in]
Filament diameter:	1.75 mm [0.069in]
Filament spool holder:	max spool width 83mm (3.25")
Number of nozzles:	2
Nozzle size:	factory default 0.4 mm [0.015in].

MAKEiT printer works with nozzle size ranging 0.25mm – 1mm.
(it is easier for smaller nozzle to get clogged)

Build platform:	heated platform made of magnetic cast aluminum
Tested printing materials:	ABS, PLA, SoftPLA, HIPS, Laywood, T-glase, Nylon
Display:	4 x 20 character LCD display
Control:	encoder with switch

MECHANICAL

Chassis:	powder coated 0.125" thick aluminum
Xyz bearings:	linear axis ball bearings
Stepper motors:	1.8° step angle
Air filter:	activated carbon filter
Extruder module:	exchangeable magnetic snap-on module
Build plate:	removable steel plate with BuildTak coating

DIMENSIONS

Product weight:	6.8 kg [15 lbs]
Product dimensions:	41L x 13W x 45H cm [16.14 x 5.12 x 17.72in]
Moving platform:	21L x 37W cm [8.27 x 14.57in]

Required front and back space for the moving platform:

Total: 62cm [24.41in]

Front: at least 27cm [10.63in] extended from the front of the metal base

Back: at least 22cm [8.66in] extended backward from the back of the metal base

SOFTWARE

Software:	open source software, "Cura" is recommended, and can be download from: http://software.ultimaker.com
File types:	STL, OBJ, DAE, AMF
Operating systems:	Windows, OSX, Linux
Connectivity:	USB and SD card

ELECTRICAL

AC input:	100–240v, 50–60 hz

Built-in power supply: 450w power output, 80 plus bronze level efficiency

OPERATING ENVIRONMENT

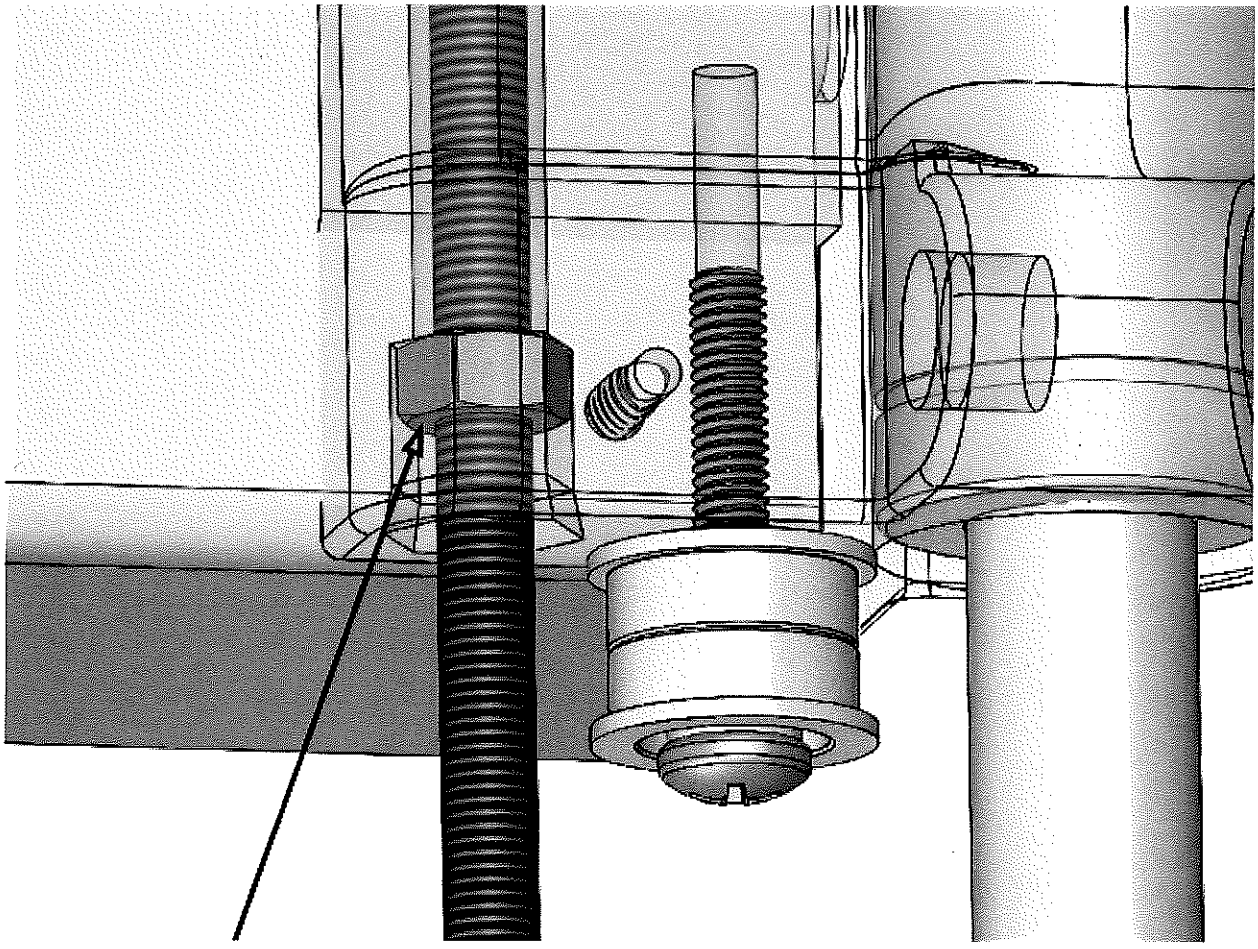
Storage temperature:	0°–32° c [32°–90° f]
Operating temperature:	15°–32° c [60°–90° f]
Humidity:	20% – 50%

2.5 OPERATIONS FLOW CHART

SETUP

3

3.1 X-BAR SETUP



On each threaded rod there is a hidden nut located inside X bar. The two nuts inside both left and right side of X bar function as lifting agents for X bar, enabling the x bar moving up or down as the threaded rod rotates. So we need to make sure both left and right side of X bar sits correctly on the nut. Otherwise X bar will

3.2 PRINTER CALIBRATION

To get the best performance from this machine, before any prints, It is recommended to follow the simple manual adjustments first, then go through the printer's auto calibration features to complete the full calibration cycle.

3.2.1 CALIBRATE HOT BED AND X-AXIS

Step 1. With the printer powered off and the build plate removed from the hot bed, use the hex screw driver provided in the toolkit to tighten the four socket screws in each corner of the hot bed. Tighten each screw all the way until it stops, then loosen each screw by four complete revolutions. With a visual check, the back end of the hot bed should appear parallel to the chassis body.

Leave the build plate aside for the time being, we will place it on the hot bed later on.

Step 2. Slide the extruder module to the far right end of the X-bar. Take note of the four semi-circular cavities in the hot bed. These are for nozzle clearance while levelling. Using a digital caliper, we need to measure the distance from the back-left nozzle clearance cavity to printer base; and the distance from back-right nozzle clearance cavity to printer base. See photo for reference.

The desired measurement between the bottom of the nozzle cavity and printer base should be 38mm. If the measurement is not 38mm, we need to either tighten or loosen the corresponding socket screw on the hot bed.

One full turn of the screw moves that corner of the hot bed up or down by 0.5mm.

Example 1: If the caliper measurement shows 37mm, we need to move the bed up 1mm. We can do this by loosening the screw, turning it counter-clockwise two complete revolutions.

Example 2: If the caliper measurement shows 40mm, the bed is 2mm higher than required distance (ie, 38mm). We can lower the bed by tightening the screw, turning it clockwise four complete revolutions.

Step 3. Switch power on. At this point the back edge of the hot bed is levelled enough so we can use the levelling feature of the MAKEiT Printer, and no need to adjust the two screws along the back edge any more.

Step 4. Press down on the control dial located next to the display LCD. This will bring us to the SETUP menu. Turn the dial to select "Calibration", and press down on the dial to choose "X-axis levelling".

MAKEiT printer aligns X bar against the back end of hot bed. All adjustments will be completed by printer itself automatically as indicated on the screen "Left and Right are levelled".

3.2.2 CALIBRATE Y-AXIS

As mentioned above, the screws along the back edge of the hot bed should not require any adjustments from this point forward. Here we will only be adjusting the screws along the front edge of the hot bed.

Step 1. This time we want to select "Y-axis leveling" under "calibration" menu.

Step 2. Upon completion of the Y-axis levelling, the printer will display two readings on the LCD screen, such as "Front left Hi: 0.25" and "Front right lo: 1.00" .These measurements (in millimeters) indicate how much to adjust the front end socket screws.

One full turn of the screw moves that particular corner of the hot bed up or down by 0.5mm.

Example 1: The display shows "Front left Hi: 0.25":
This means front-left corner is 0.25mm higher than the back-left corner.
Solution: We tighten the front-left screw by turning it clockwise for one half revolution.
Example 2: The display shows "Front right Lo: 1.00":
This means front-right corner is 1mm lower than back-right corner.
Solution: We loosen the front-right screw by turning it counter-clockwise for two full revolutions.

After the above adjustments, we need to run "Y-axis levelling" from the SETUP menu again. Once the Y-axis is properly leveled, we will see the following message on the printer screen:

Left Edge Leveled
Right Edge Leveled
Click to Proceed

If the display shows new readings are within +- 0.03mm range, then Y-axis is well leveled.

3.2.3. CALIBRATE NOZZLE DISTANCE

This is also referred as nozzle height adjustment.

Step 1. Put black build plate onto hot bed. Make sure there is no plastic or any other debris between build plate and the heated bed.

Step 2. Place a piece of clean regular copy paper on the build plate (regular copy paper is 0.1mm thick).

Step 3. At printer's "SETUP" menu, select "Calibration", then choose "Nozzle distance" command. Once extruder module moves to the center of the build plate and stops, let's hold the paper gently and swing it side to side,(shown pic) We may encounter a few possible scenarios:

Scenario 1: Only the left nozzle touches the paper. Especially when we swing the paper from side to side, we can feel that swing rotation is focused on the tip of left nozzle. This tells us the right side nozzle is higher than the left size nozzle.
Solution: To bring down the right nozzle, we can use the hex driver in the tool kit to loosen (counter clockwise) the screw located inside top right corner of the extruder module. (shown pic)

Scenario 2: Only the right nozzle touches the paper. When we move the paper from side to side, we only notice the rotation focus point is concentrated on the tip of right nozzle. This shows left nozzle is higher than right nozzle.

Solution: To bring down the left nozzle, we can use the hex driver to tighten (clockwise) the screw inside the extruder module.

Scenario 3: Both nozzles touch down on the paper, we sense an distorted focus point generated by both nozzles. This is the correct nozzle position to start with, since both nozzles are levelled horizontally.

Once the two nozzles are levelled, the next thing is to set the nozzle distance manually. To start, let's try sliding the paper gently back and forth to sense the dual nozzle frictions. We may encounter a few different scenarios:

Scenario 1: We find the copy paper is locked in place and hard to move. This indicates that the nozzles are too close to the build plate. This may cause our printed parts stick too well onto the build plate, extreme difficult to be removed. It also may put undue strain on the feeder motors as the filaments cannot flow properly.

Solution: We can turn the printer's control dial clockwise slowly to raise the extruder module up a little until the copy paper can be slipped back and forth, while retaining some level of resistance.

Scenario 2: If we don't feel any friction at all while moving the paper, then the two nozzles are too far from the build plate. The first layer of the print likely will not stick to the build plate, causing an unsuccessful print.

Solution: We need to turn printer's control dial counter clockwise to bring the extruder module down until both nozzles touch down the paper. The sweet spot is somewhere we can sense the nozzle friction and the copy paper can be still slided back and forth.

Besides nozzle distance, the filament quality can also affect the first layer quality/stickiness.

We should re-calibrate the machine after any event that may place outside physical forces on the machine, such as moving or traveling, changing nozzles, changing guide tubes. It is also a good practice to calibrate after switching different filaments.

3.3 LOADING FILAMENTS

The procedure to load or change filaments for dual nozzles is the same as the procedure to load filament into single nozzle. When viewing the machine from the front, Nozzle 1 (marked 1 on the top left filter cover) is on the left side of the extruder module, the corresponding filament feed, Feeder 1, is on the right side of the X-bar. Nozzle 2 is on the right side of the extruder module, while the corresponding Feeder 2 is on the left of the X-bar. It is important that the filaments properly cross over in an X pattern, otherwise the dual printing may not function properly.

3.4 SOFTWARE SETUP

The software we recommend for use with the MAKEiT printer is Cura, a well-documented software package produced by Ultimaker. To download the newest version of Cura, go to:
http://software.ultimaker.com/
We should install Cura onto our computer, but do not start the program until we update some setup files to work properly with the MAKEiT machine.

NOTICE: DO NOT START CURA BEFORE FOLLOWING THE STEPS BELOW.
Some default settings in Cura do not work with MAKEiT printer, so we need to be careful to make the proper software adjustments before attempting to print. If the default settings are not properly adjusted it may cause the following issue: The printer's extruder module may attempt to operate outside the build plate area, or lower the nozzle position below the level of the build plate surface. This can cause malfunction and possible damage to the build plate and nozzle. Thus, it is vital to integrate MAKEiT setup data in Cura first before we run the program.

Located on the included SD card is a settings file labelled "MAKEiT Pro Machine Settings.ini". This file allows Cura to successfully communicate with our printer. To integrate the MAKEiT machine settings we can follow these steps:

For MAC user:
Open Applications folder, right click on Cura folder
Go to "Show Package Content"
Click on "Resource"
Open "Machine Profile" folder
Find and copy "MAKEiT Pro Machine Settings.ini" file located on our SD card
Paste this file into Machine Profiles folder.

For PC user:
Find "MAKEiT Pro Machine Settings.ini" file on our SD card
Copy this file
Paste this file into C:\Program Files (x86)\Cura_###\resources\machine_profiles (where ### will be replaced by the version number of your Cura installation, for example Cura_14.09 at the current time of writing. If you chose a custom installation folder when installing Cura, your folder directory will be different)

Initializing Cura for the first time brings up a setup dialog with a few pre-configured printer settings. At the first dialog, choose "other", in the second dialog that follows we should see "MAKEiT Pro" available as an option. Alternatively, if we have already started Cura and setup another printer, we can follow these steps to add our MAKEiT printer.
Add "MAKEiT Pro" machine
Click on "Machine Setting"
Choose "Add New"
Click on "Next"
Select "Other" and go "Next"
Click on "MAKEiT Pro Machine Settings" and go "Next"
Click "Finish" button
Click "Ok" under "Machine Setting" window

Now we are ready for printing!

4 PRINT

4.1 BASIC SINGLE-NOZZLE PRINTING

Printing from SD card is the more reliable and adjustable method for printing. Using SD printing gives us more control over the print by allowing us to adjust a greater number of settings during a print, settings which are locked by the PC software if printing via USB. Also printing via SD allows us to shut off our computer during long prints.

Step 1: First we import our .stl, .obj, or other file type into Cura and adjust desired settings accordingly

Step 2: Insert the including USB/SD combination card into the USB port of your computer

Step 3: Once inserted, we click the save icon in the top-left of Cura's graphic interface (Icon is initially shaped like a 3.5" floppy disk, but will change to the shape of an SD card and bear the label "SD" after inserting our card). This will save a .gcode file onto SD with the original filename from the imported part ("part1.stl" gets saved as "part1.gcode")

Step 4: Alternatively, if we wish to specify the name of the .gcode file that gets created, we can use the "File" dropdown menu on the top menu bar and select "Save .gcode". This will bring us a save dialog where we can specify the filename and path

Step 5: Now we properly eject the SD using the proper method from Windows or MacOS, and insert the card into the front slot of the MAKEiT machine

Step 6: Press down on the silver encoder knob located next to the LCD screen to bring up the "SETUP" menu.

Step 7: Turn the knob to highlight "SD PRINTING", and press down on the encoder knob to select

Step 8: We are now presented with a list of files contained on the SD; the most recently saved file should appear at the top of the list. We use the encoder knob once again to highlight and select our desired file. After selecting the file, the printer will heat up to the preset temperatures and our print will start

4.2 ADVANCED PRINT USING DUAL NOZZLES

The variability of dual nozzles gives us the option to use the printer a number of different ways to serve the needs of our print. We can use filaments of the same material in similar or different colors to duplicate prints for batch production, or we can use different color filaments to print one item in two color patterns, or we can use two different materials one for the part and one for support (such as ABS with HIPS). The first thing to do before we print with dual nozzles is to double-check we have both nozzles properly loaded with the desired filaments.

4.2.1. ONE MODEL IN TWO COLORS OR MATERIALS

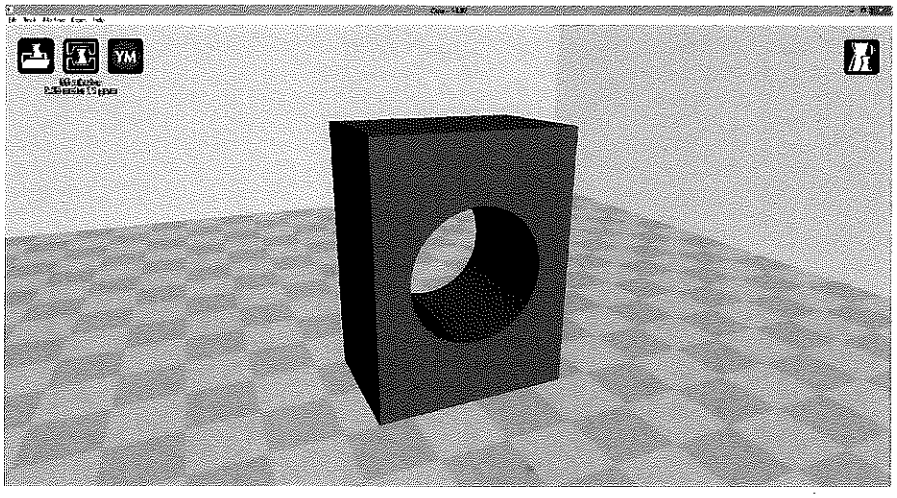
Step 1: Properly export the design files: When preparing an object for printing with two colors or different materials, we should design it in our CAD software as a single file with two separate bodies. From the single file, we then export the different color/material as separate .STL files. For example, we have a two-color blue and red object (one CAD file) as shown in the picture. From our CAD software we will save/export the red portion as "RedPart.STL", and blue portion as "BluePart.STL". If these two separate .STL files are exported from the same CAD file, Cura will be able to properly realign them.

Step 2: In Cura under "File" tab, select "Preference" to choose the desired colors for two extruders. This is only for visual preference in Cura (the color represented on your screen does not change the filament you have loaded in the printer..

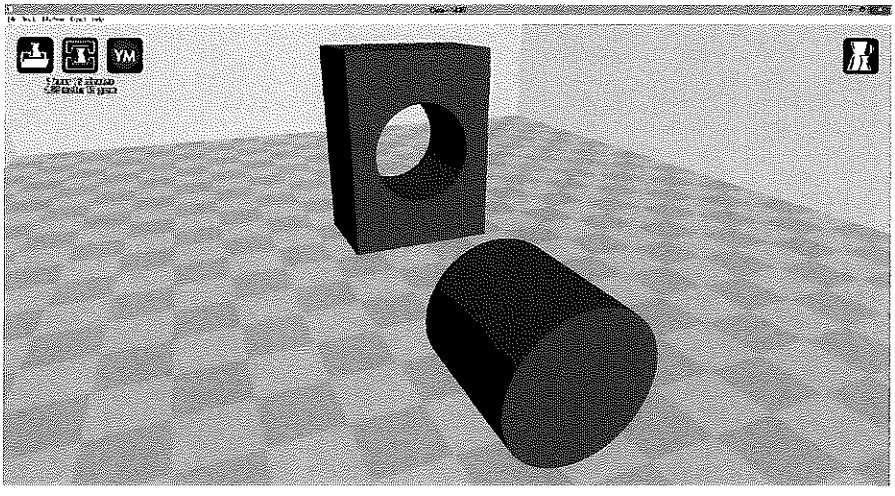
Step 3: Properly designate materials for the two different parts: The .STL file that is loaded first will be assigned to print from Nozzle 1, the file loaded second will be assigned to print from Nozzle 2.

Step 4: In this case, we have loaded blue filament into Nozzle one, thus we first load the file "BluePart.STL". With red filament loaded into Nozzle 2, we correspondingly load the part "RedPart.STL" second. Make sure the colors/materials of the filaments on your printer are set accordingly.

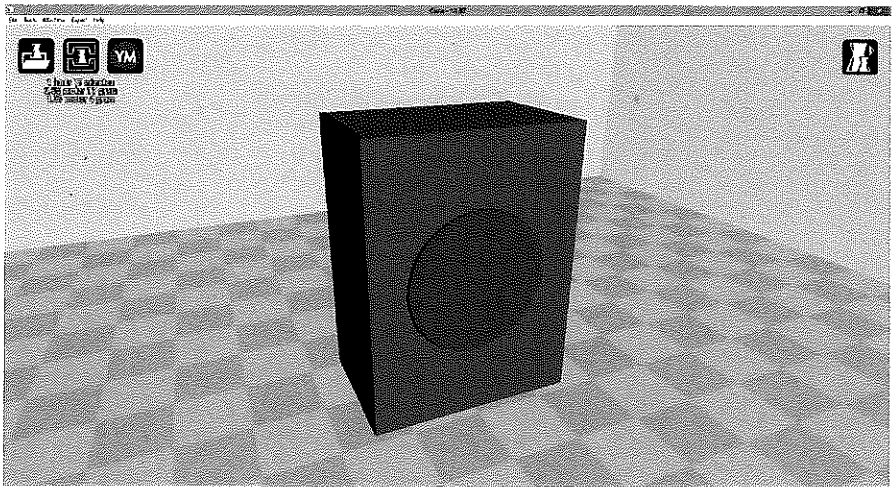
Step 5: Load .STL files in Cura ; BluePart.STL is loaded first on Cura bed.



Then the cylinder file is added to Cura bed.



Step 6: Dual Extrusion Merge: Once the "BluePart.STL" and "RedPart. STL" are loaded in Cura bed, we right click on either item to bring up a dialogue window. When we select "Dual Extrusion Merge" the two models should merge automatically back to their arrangement from our original design in our CAD software. The object on Cura bed is now shown in two different colors.



For optimal prints, we need to make setting changes in each of the Basic, Advanced and Expert menus

Basic Menu

2nd Nozzle Temp (under "Speed and Temperature"): enter "0". Printer will assume 2nd nozzle temp the same as the 1st nozzle.

Support Dual Extrusion (under "Support"): enter extruder /nozzle number. This is the extruder / nozzle we want to use to print support material. Usually extruder 1 is for primary building, extruder 2 is for support.

Diameter and Diameter 2 (under "Filament"): If possible, we should use a caliper to measure the exact diameter of each filament, though sometimes it may be safe to assume an expected diameter of 1.75mm. Enter "1.75" for "Diameter", if your second filament is the same type as the first, you can enter "0" for "Diameter2". If you have measured a different diameter for your second filament, you may specify it here.

Advanced Menu

Retraction:
Speed (mm/s): 100
Distance (mm): 4.5
Dual Extrusion Switch Amount (mm): 15
Cool: enable cooling fan to prevent dripping of material to the printed parts

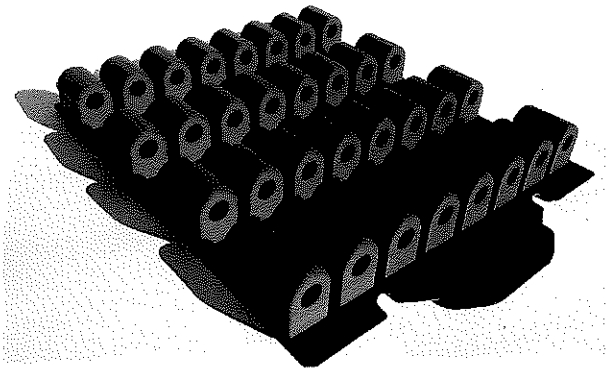
Expert Menu (click on "Expert" tab and "switch to full setting")

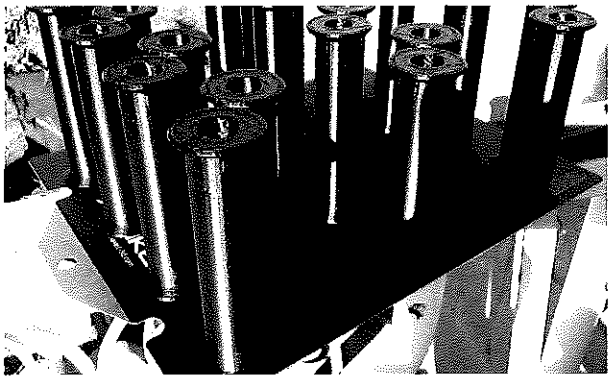
Z hop when retracting" under "Retraction": usually it's set around "0.3 - 0.5", we use "0.3"mm often.
Fan speed: min 50% max 75% for ABS; 100% for PLA

We are now ready to run our first dual-headed print!

4.2.2. DIGITAL BATCH PRODUCTION

It is usually rare to use a 3D printer for quantity productions due to slow speed and lack of precision. However the unique features of our MAKEiT printer allow for digital production in batches of meaningful quantity and uncompromised quality. Here is an example shown below. We at MAKEiT frequently use the MAKEiT printers to produce multiple parts in once.





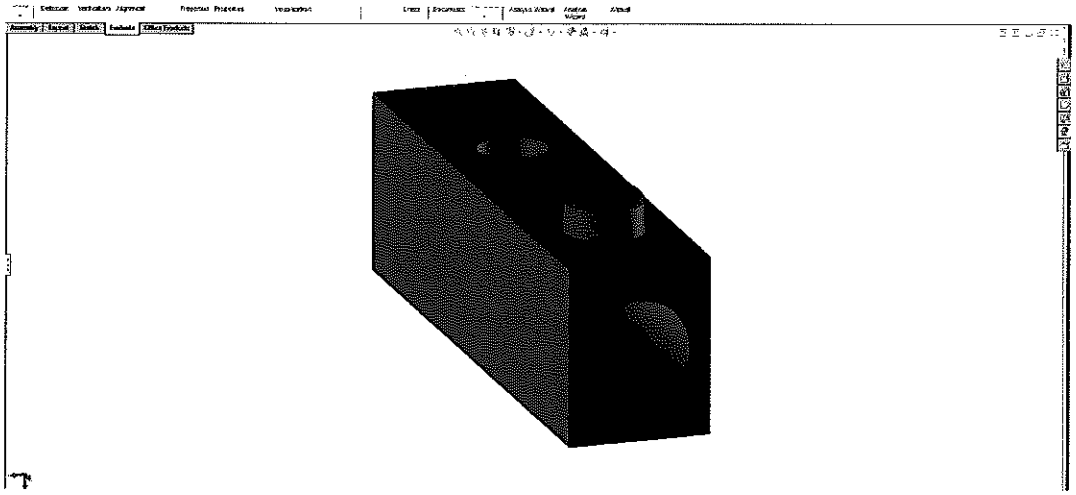
In the first picture 32 parts were printed on one build plate by one printer in 8 hours. These end parts are enough for 8 MAKEiT printers. In the second picture 24 reel holders were produced by one printer in one production run.
It is this duplication and simple production process that opens the door to future digital batch manufacturing.

Enabling these time-effective batch prints is the printer's key function of "Duplication". Our MAKEiT printer is capable of duplicating individual parts to make entire batch prints. The only limitation is that the copied item's width is within 48mm (due to the fact that the distance between two nozzles is 50mm.)
The duplication printing sequence depends on the specific machine setting we input at the beginning of Cura setup. Safe way to start is to print from front row to back row. The Tool Path display option in Cura is a very good indicator to check first before we hit the print/save button. Tool Path shows us in detail where and how our nozzles extrude filaments. Below is a step-by-step instruction for duplication, using a printer part example.

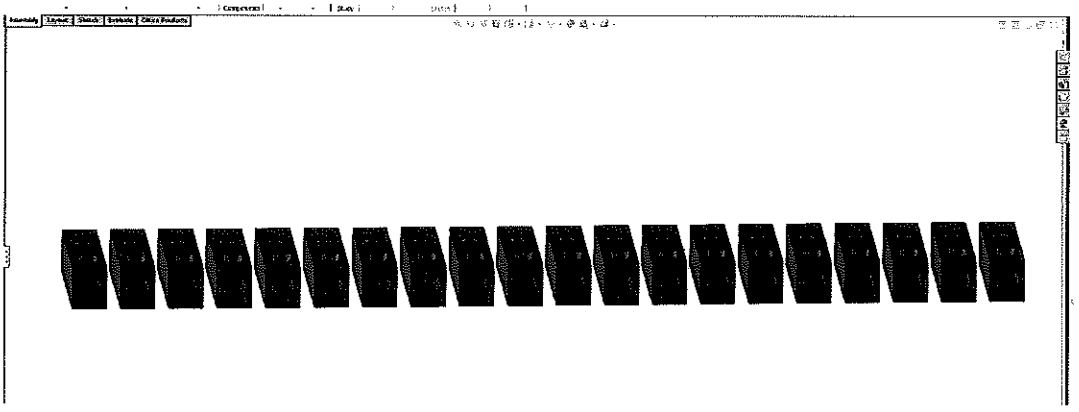
4.2.2.1. PREPARING DUPLICATED ITEMS

We will use a printer part (30mm long 7mm wide) as an example.

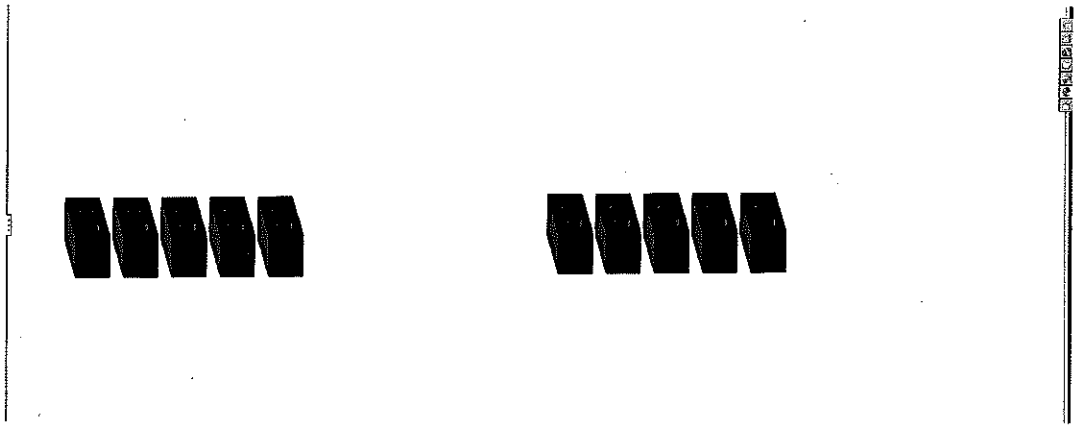
Step 1. Preparation In design software
This example uses SolidWorks as our CAD software, but similar functions are available in most CAD softwares and this method should be adaptable.



We will use "Linear Component & Pattern Direction" features under "Assembly" window in our design software to make the proper layout. A proper pattern (sometimes also called an "array") is set up as follows.
Since our build plate is 200mm wide, the distance between two nozzles is 50mm, and the width of the part is 7mm, so to figure out how many possible parts to print in one row, we need to calculate as this:
50mm / (part width + gaps between two parts) = 50mm /(7mm+1mm) = 6pcs approximately.
Since we will have 4 columns (200mm wide / 50mm nozzle distance) in a plate, The number of parts we can possibly print is 6x4 = 24pcs. So a row of 20 parts across the width of the build plate is appropriate.



Our assembly components should now appear aligned as above. To make use of the duplication feature, we need to remove the parts from the array that will be printed by the second nozzle. Counting from the left, in our case we keep the first 5 items, delete the 6th, 7th, 8th, 9th, and 10th items; keep the 11th through 15th item, delete the 16th, 17th, 18th, 19th, and 20th items.



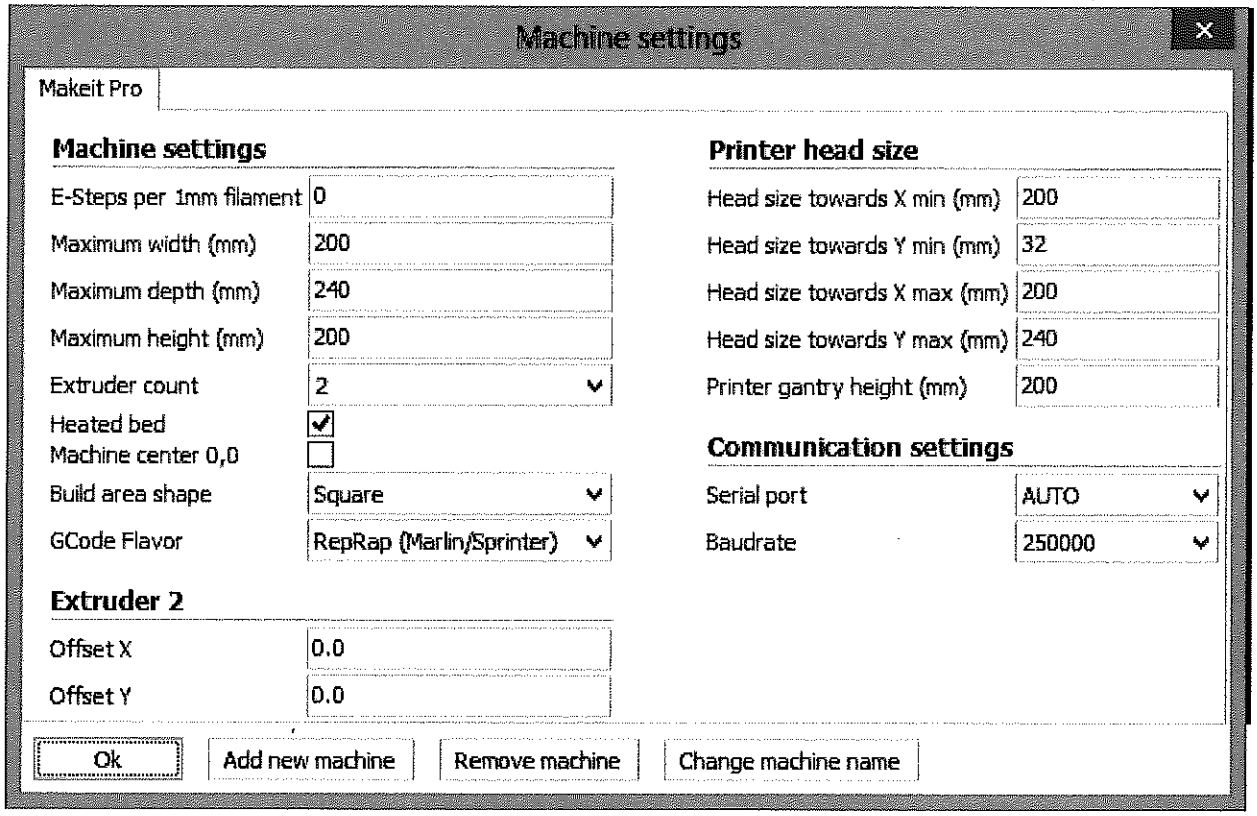
Now we have two columns, 5 items in each column. Why erasing some of the items? We will explain in Step 2.4.

Save this whole assembly of parts with a new filename, then export it all as a single .STL file.

Please note, it is important that we export our multiplied parts as a single .STL file, so that the slicing and printing software Cura will treat it as one object and take action accordingly. Otherwise we will not see the full benefits of duplication features.

4.2.2.2 PREPARING CURA FOR BATCH PRINTING

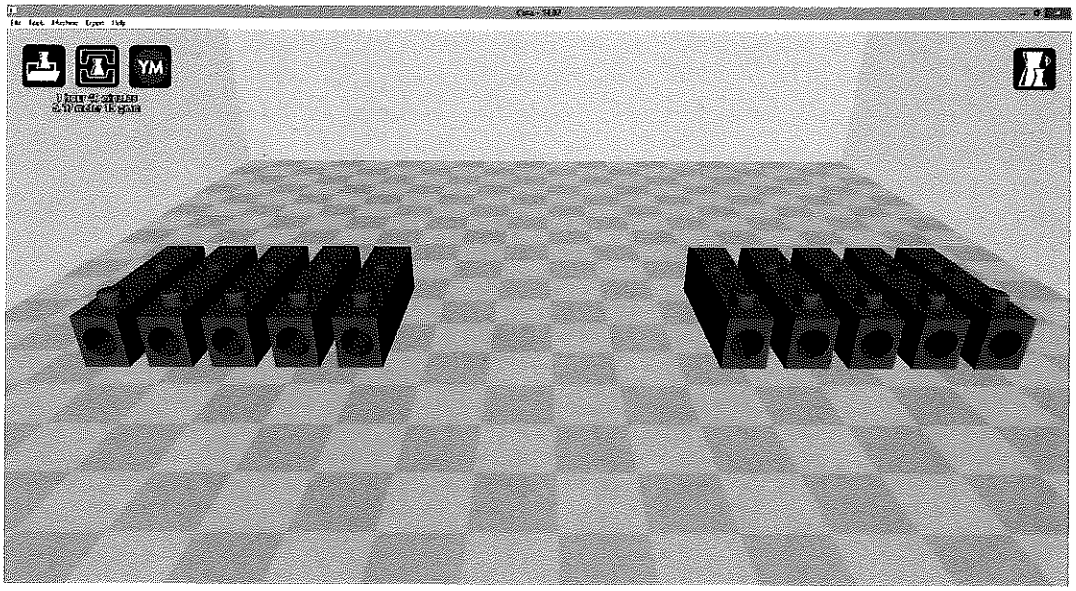
1. Make sure "Machine Setting" in Cura is set as shown below:



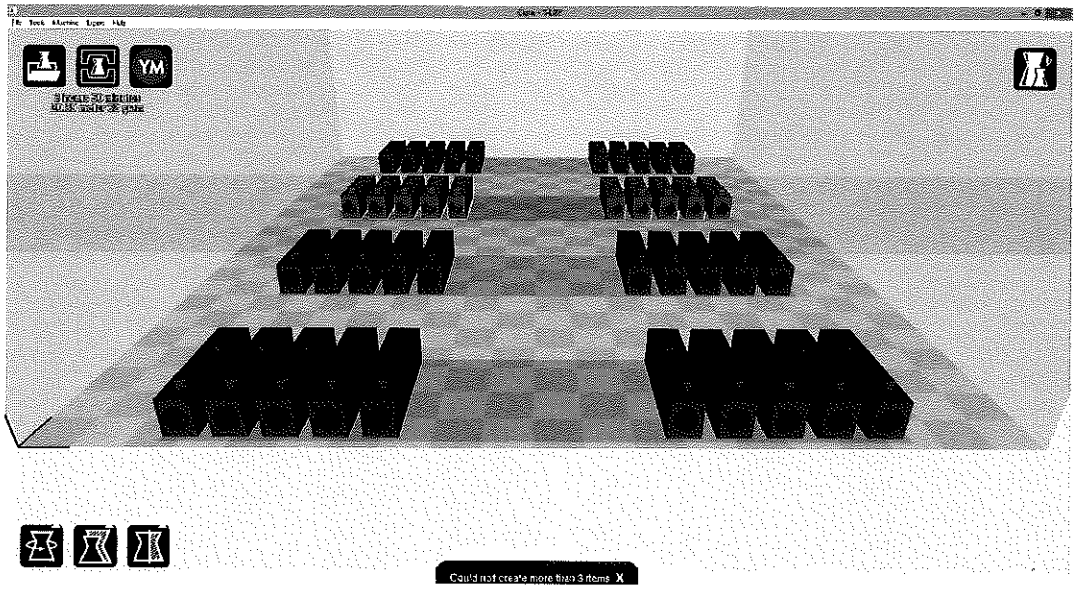
*Head size towards Y min 22mm: When printing with PLA material, we may remove the filter and filter cover from the extruder module. In such case, the distance between two rows on Cura bed should be kept at least 22mm apart, ie, at least two checker boxes away. Otherwise, the printed item will be knocked by the extruder module as it moves to the next row of printing.

*Head size towards Y min 32mm: When printing with ABS material, it is advisable to have the filter and filter cover attached to the extruder module. The distance between two rows on Cura bed should be kept at least 32mm apart, or three and half checker boxes away. Otherwise the printed item will be knocked by the extruder module's filter cover as it moves to the next row of printing.

Load the printer part STL file into Cura's bed.



Right-click on any object to bring up a popup window. Choose "Multiply object" and input the number of parts desired.

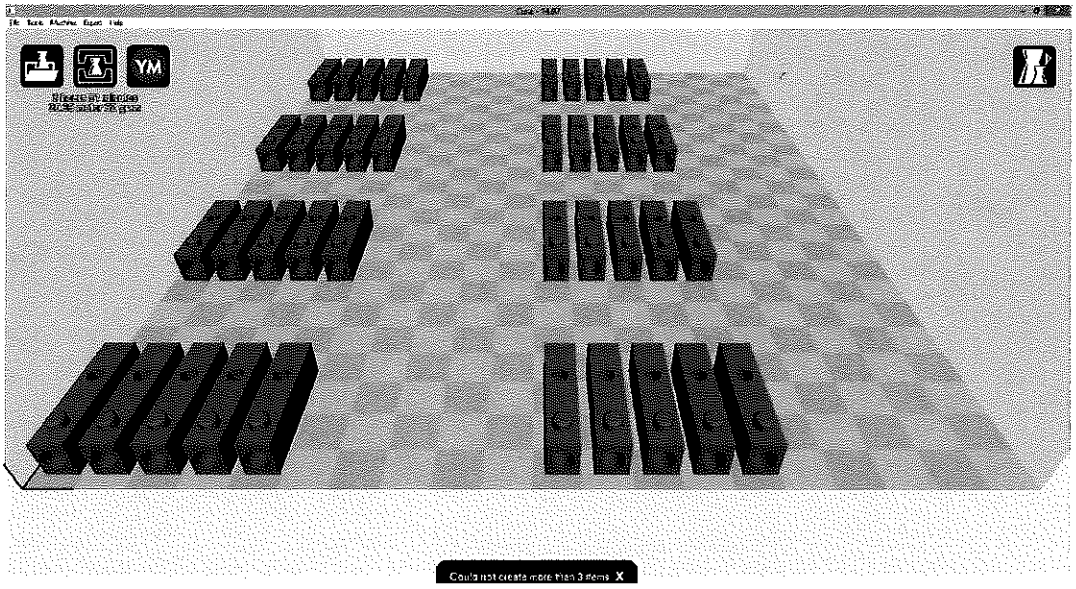


On Cura screen we see two columns visible with multiple rows. Actually, it is one visible column followed by an invisible column, so there are total four columns of parts to be printed in this particular file.

Why do we erase some of the items earlier in the design software? We do this to allow clearance for Nozzle 2 to do the work of printing those copies which we removed. Nozzle 2 will move in the same locations where 6th-10th and 16th-20ths items are, and will print them simultaneously. Even though they don't appear in Cura, these items will be printed when we select the "Duplicate" feature on our printer.

At this stage we need to make adjustments so that the last invisible column at far right will have enough space on the build plate and can be printed well.
As a rule of thumb, each checker box represented in Cura's software is 10mm by 10mm. Start from the right edge and count 5 checkers (or 50mm) to the left, this will be the starting place to print the second invisible column. If any item appears as dark grey in Cura, it means there is not enough space for that item to be printed.

Now we need to properly arrange our rows of parts in the Y-axis, front to back. As we spread the multiplied rows out, we should take care to leave the proper amount of spacing between them.



Choose all the necessary settings in "Basic", "Advanced" and "Expert" menu in Cura.
Select the "Tools" drop-down menu from the Cura window, then select "Print one at a time" command. This will set the printer to finish the first front row (which we actually loaded as one part) before moving to the next row.
Under "Start/End GCode" window, highlight "Start GCode" to make sure gcode M823 (activate duplication from PC) is available in the list of commands. If not, we need to add "M823 ; activate duplication" directly above the line that reads "G28 SC". Alternately, we can also activate duplication manually, directly from the setup menu on the printer.

Save the GCode file

4.3 SD PRINTING AND MANUAL DUPLICATION OPTION

If we chose not to add the "M823" line to the "start.gcode" file we can still set duplication manually on the printer. After completing the preparation work for our print (4.2.2.1), bring down the "File" menu and choose "Save GCode", or use the shortcut on the screen to save GCode to SD card.

1. Insert SD card with print file into the printer.
2. On the printer, bring up the SETUP menu by pressing down on the encoder button, and select "Duplication". (Remember, this step is only for if we chose not to set duplication in Cura using the "M823" line. If we HAVE done preparation with Gcode M823 in Cura, then we don't need to choose this selection.)
3. Select SD printing
3. Locate the desired file utilizing the encoder dial
4. Press down on the encoder to select the particular file
5. Print will start once the printer is warmed up

If we desire a large quantity batch with more than one print we can do so quickly with multiple print plates. After the first run of 20 parts are printed on the print plate, we can take this full sheet off the hot bed (with proper heat protecting gloves) and put on a new built plate. Simply choose SD Printing again and choose your GCode file. The duplication process starts once more, another 20 parts can be produced easily.

4.4 ON-THE-FLY TUNING DURING PRINT

After we load our STL file in Cura, it is required to input numerous variables, such as speed, temperature, infill, flow rate, etc. into Cura's settings before it can "slice" the STL to produce the GCode for MAKEiT printer to print the file.

These settings are given based on our earlier printing experience, or just our pure guessing sometimes. Once the print job begins all of these values are fixed and cannot be changed through Cura, unless we cancel the print in order to change the values.

This is not convenient for us. To have a better printing experience, our MAKEiT printer provides a series of on-the-fly adjustment functions, that are grouped under TUNE menu. This allows us to correct or fine tune the values we set earlier in Cura.

At any time during a print we can rotate the encoder dial in either direction to access the TUNE menu. Any changes made here happen in real-time for the current print, giving us the flexibility to make adjustments in key areas such as fan speed, print speed, nozzle temperature, bed temperature, flow rate, and nozzle height, without the time cost of restarting the print.

Here are some examples of fine tune functions.

-Nozzle temp
When we see whisker on the printed object, we can raise the nozzle temperature 5c higher. Each nozzles' temperature can be adjusted individually.

-Bed temp
If some of the first layer print does not stick well, we can increase bed temp by 5c incrementally after we finish nozzle height adjustment. Usually bed temperature settings are between 80c-90c, occasionally 95c depending on the building material and how much contacting area our model has on the bed. If the object has small contact area, we tend to raise bed temp a little higher; if object has large contact area, we can bring the temp down a little for easier removal once the print is finished.

-Fan speed
When we build single object in small size, for example, a M4 screw, we can increase fan speed.
For PLA printing we always run the fan speed to max.
For dual PLA printing it is recommended to have fan on full speed.

-Printing speed
Depends on our need, we can speed up the printing time if resolution is not the highest priority, or we can slow down the printing speed to have a finer, higher-quality part. We may explore to find the right speed for certain print, or certain materials. It is also possible to adjust speed for different portions of a single part, if we have variable levels of detail in the same part.

-Flow rate
Flow rate controls how much filament is extruded and is extremely import for a successful print. When too much filament is extruded within certain time, we will notice things like bulging area, bubbles, droplets, or extra grooves, causing our print surface uneven. If too little filament is extruded, the outer walls can be too thin, the infill may not properly connect with the walls, gaps may be found between each layer, or yet other defects. Being able to adjust the flow rate on-the-fly, we can correct or re-enforce our print.

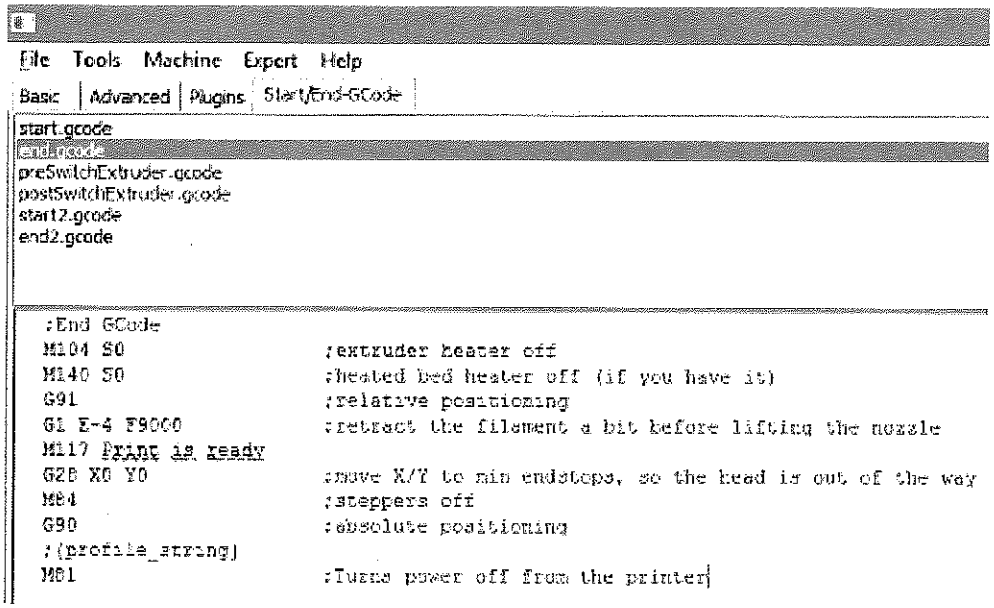
-Nozzle Distance
In addition to the nozzle calibration at the setup time, we have the opportunity to fine tune the nozzle height at any time when it is necessary by selecting "Nozzle distance" at "TUNE" menu. This feature enables us to double-check for proper nozzle distance at the beginning of a print to ensure we have a good solid first layer foundation.

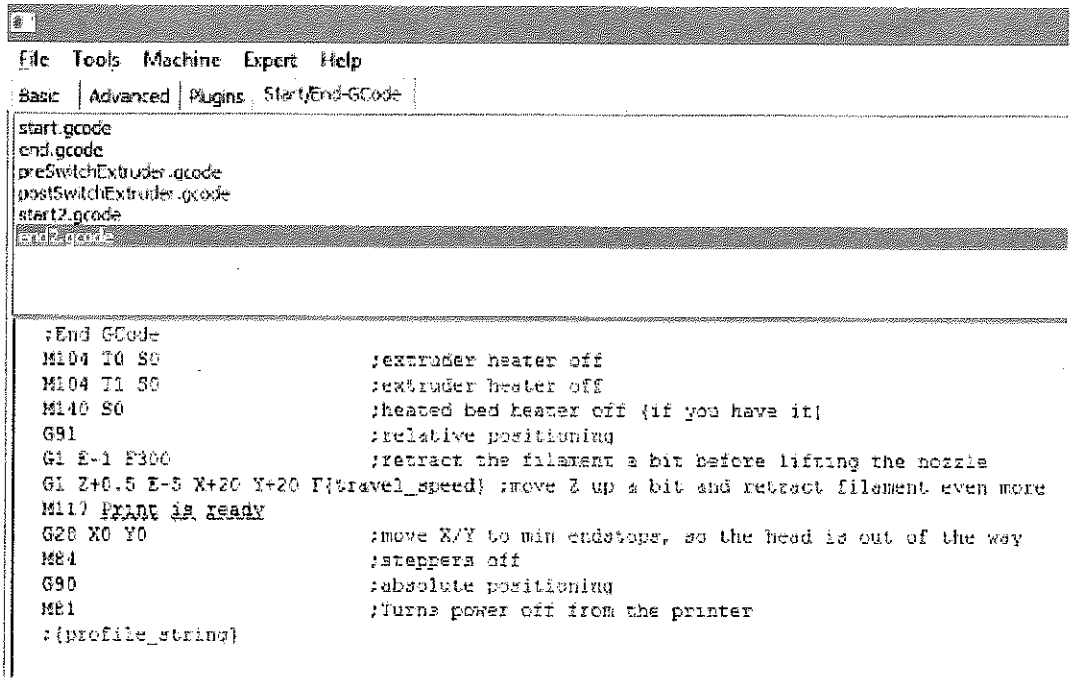
We can also utilize the "brim" or "skirt" feature in Cura to check out the right nozzle distance. Since brim or skirt is printed first before the real object, we can use them to see how the first layers are extruded, then easily adjust the nozzle distance accordingly before the actual part starts printing.

4.5 FINISHING A PRINT WITH AUTO-SHUTDOWN

Once a printing job is done, the extruder module will move back to the home position. We may keep the printer on for another print job or switch it off manually. For convenience, especially with overnight printing, our MAKEiT printer has an auto-shutdown feature to power off automatically after the job is done.

To set up the auto-shutdown option, we need to add a line of code to the end.gcode file in Cura:
Under "Start/End GCode" menu:
For single nozzle printing, click on the "end.gcode" file
Add "M81" as the last line by typing in the bottom field;
For dual nozzles printing, repeat the above step for file labeled "end2.gcode"





5 MODEL REMOVAL AND FINISHING

5.1 REMOVING MODELS FROM BUILD PLATE

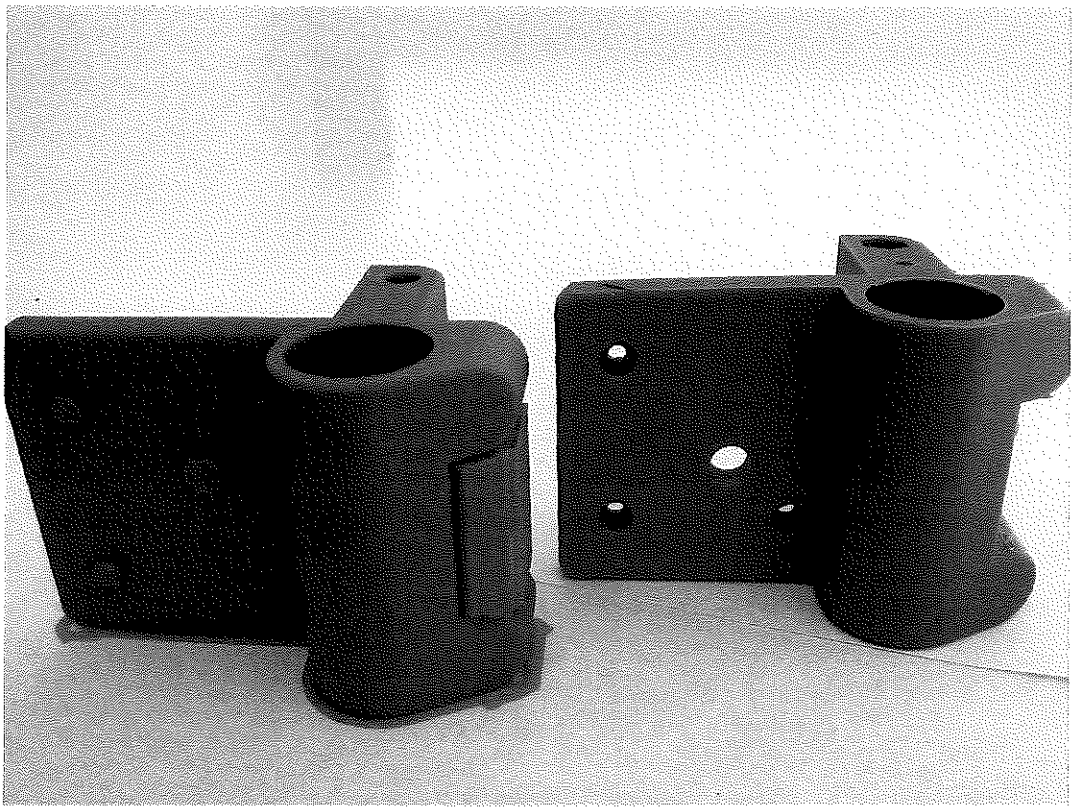
When we have a plateful of items, items that are densely built or large in size, it is recommended to remove the build plate from the printer while it is still warm, being sure to use the safety gloves provided. We can then utilize the putty knife provided in the tool kit to gently lift up the prints, little by little.

If the plate is allowed to cool before removal, sometimes it can be additionally difficult to remove our prints from the cold build plate, as the prints can be stuck onto the plate quite firmly. In this case, strong use of the putty knife may damage our print or the build plate. In this situation, it's better to heat up the plate on the hot bed to the same temperature used to print these objects before removal. It will then be easier to remove our printed objects off the plate without unnecessary wear.

5.1 REMOVING MODELS FROM BUILD PLATE

Depending on the object's shape and geometric structure, supporting materials may be printed and bonded to the object itself.

Here is a picture of MAKEiT printer parts on the build plate. These four parts are built with supporting materials, indicated in the above image by orange labels. These supporting attachments are thinly built and can be easily broken away by hand, or removed with a handy tool. When support structure is printed with the same material as our actual part, no liquid solutions of any kind are necessary to clean up these parts. After the supporting materials are peeled off cavities are revealed, as shown below. Now these parts are ready to be installed on the next MAKEiT printer.

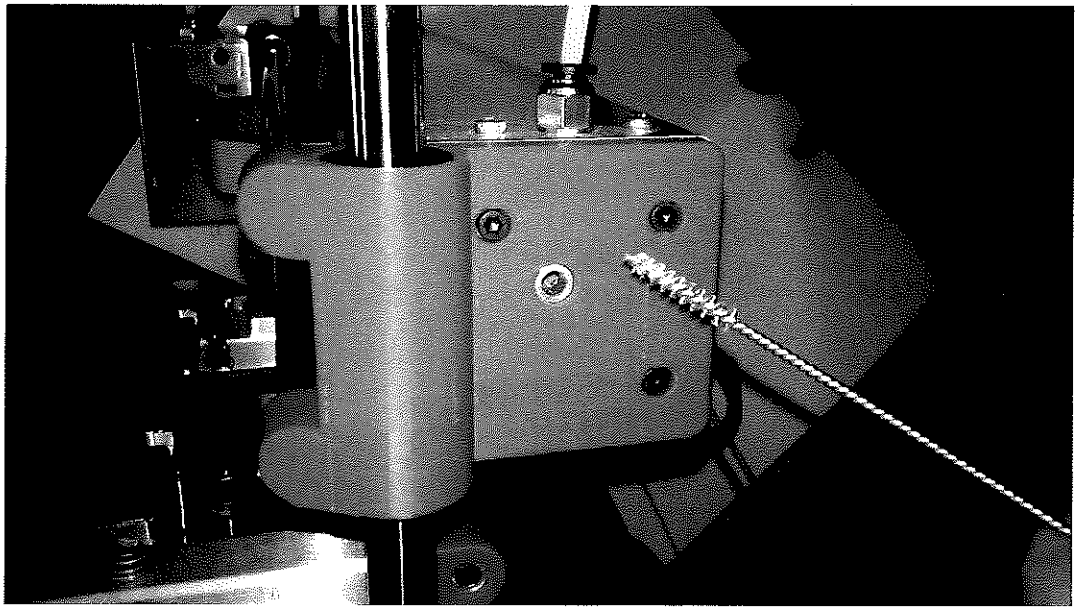


6 MAINTENANCE

No matter the brand, all 3D printers have moving parts and thus require a certain level of maintenance. The MAKEiT printer was designed to make maintenance activities as easy and convenient as possible, to ensure we get the most out of our investment. Regular care of our MAKEiT printer will not only improve the quality of prints, but also extend the life of wearable components in the printer.

6.1 CLEANING THE FILAMENT FEEDERS

There are two feeders inside X Bar. We can access these filament feeders for cleaning through the tunnels located on both left and right end of the X-bar using a metal brush.



6.1.1 CLEAN FEEDER 1

Step 1: Press down the control dial to bring up the SETUP menu.
Step 2: Select "Filament", then select "Filament 1"
Step 3: Choose "Move filament"
Step 4: Turn the control dial in either direction to move filament 5mm.
Step 5: Once the filament has stopped moving, insert the cleaning brush (provided in the toolkit) into the cleaning tunnel. Move the brush back and forth a few times to clean the feeder.
Step 6: Remove brush from the cleaning tunnel. DO NOT ATTEMPT TO MOVE THE FILAMENT FEEDER WHILE BRUSH IS INSERTED!
Step 7: Repeat steps #4, #5 and #6 a total of 8 times to complete cleaning the feeder.

6.1.2 CLEAN FEEDER 2

Press down the control dial to bring up the SETUP menu.
Select "Filament", then select"Filament 2"
Choose "Move filament"
Turn the control dial in either direction to move filament 5mm.
Once the filament has stopped moving, insert the cleaning brush (provided in the toolkit) into the cleaning tunnel. Move the brush back and forth a few times to clean the feeder.
Remove brush from the cleaning tunnel. DO NOT ATTEMPT TO MOVE THE FILAMENT FEEDER WHILE BRUSH IS INSERTED!
Repeat steps #4, #5 and #6 a total of 8 times to complete cleaning the feeder.

6.2 LUBRICATING XYZ AXES

It is recommended to lubricate all rods on X,Y and Z stages every two - three months. A lubricant stick is provided in the tool kit. Apply thin layer of it on all rods, including the two Y direction rods under the hot bed.

6.3 REMOVING AND REPLACING NOZZLES

1. Select "Maintenance" from the printer's LCD display menu.
2. Select "Nozzle Exchange"
3. Printer will automatically heat up the nozzle and raise the extruder module to the proper height for easy access.
4. Wait until LCD screen's prompt when it's ready to proceed.
5. Be sure to wear safety gloves and use hex nut driver included with the toolkit to take out the hot nozzle.
6. Using the same hex driver and still wearing the safety gloves we can now screw in the new nozzle.
7. After replacing old nozzle with new one, press down the control dial, and the extruder will go back to the home position.
8. Complete the nozzle exchange by recalibrating the nozzle height, from the calibration menu.

6.3 AUTOMATIC NOZZLE CLEANING

The MAKEiT printer has the ability to clean away any risidual filaments on the nozzle, utilizing the built-in metal brush. We can do this by pre-programing G-code in Cura:

. In the Cura window, select the tab "START/END GCode"
2. Select the "start.gcode" file
3. Where you see "G28" in the code, change it to "G28 SC".
4. Save the GCode.

6.4 MANUAL CLEANING FOR CLOGGED NOZZLES

First, we follow the steps from 6.3.1 again in order to remove the nozzle for cleaning. After removal, we can either soak the ABS-clogged nozzle in acetone liquid (it may take quite some time to dissolve ABS completely), or burn out the old ABS/PLA debris on hot flames.

NOTICE: Take care when using blow torch, stove, or other high source; safety gloves and a long handle plier are needed to prevent any injury.

Once nozzle is cleaned with either heat or chemical, we can finally use a 0.4mm drill bit (or properly sized for your alternate nozzle) to ensure the hole is clear.

Lastly, we reinstall the cleaned nozzle and once again run the nozzle levelling procedure.

Other nozzle cleaning methods can be found on internet, user discretion is advised.

Professional service and maintenance can be arranged to save user's time. MAKEiT Inc. is proud to offer local users in Southern California extra services such as customized 3D design and machine solution, on-site maintenance and networking.

TIPS, TRICKS, AND TROUBLESHOOTING

4

4.1 GENERAL TEMPERATURE GUIDELINES

ABS
Nozzle: 210-250 C
Bed: 80-100 C
PLA
Nozzle: 180-220 C
Bed: 50-60 C

Temperature settings vary for different materials, and . It is highly recommended that we practice with the existing material and use TUNE feature to find the right temperature.

4.2 TRICKS FOR PRINTING WITH PLA

1. Since we don't need to filter PLA fume, we can take filter and filter cover out. This gives extra airflow needed for PLA printing.
2. Increase fan speed to the maximum so that more air flows through the extruder.

4.3 TRICKS FOR SHELL/WALL THICKNESS IN CURA

The shell/wall thickness should be set at multiples of nozzle size.
For an example, our printer's nozzle size is 0.4mm, wall thickness should be set at
1x0.4mm = 0.4mm;
or 2x0.4mm =0.8mm;
or 3x0.4mm =1.2mm;
or 4x0.4mm=1.6mm, etc.
If it's not the multiples of nozzle size, for an example, if we set at 0.9mm, Cura will only use 0.8mm, our print will be made with 0.8mm wall thickness, NOT 0.9mm.

4.4 TRICKS FOR OVERHANG SUPPORT

1. At Cura "Basic" window, select "everywhere"
2. Pull down "Expert" menu, input "0" at "support overhang (degree)": the smaller angle, the more support under overhang part.
3. Set "Distance X/Y mm" at 0.7mm. This covers all area of overhang.
4. Set "Distance z mm" at 0.15mm
5. Set "Fill amount" at 15%

Tip: use "Visual" button on the top right of Cura screen to see how the support material is printed before hit "Print" button.

4.5 Q & A FOR COMMON PRINT TROUBLES

What to do when printer stops in the middle of print?
This can happen during PC printing. Sometimes computer goes into "sleep" mode, causes printer stop printing temporarily. We need to make sure to turn off "sleep" mode on PC.

How come my printer can not link with Cura?
Sometimes the communication link is missing. We need to re-set the link by following the steps here:
1. Select "Machine setting" in Cura
2. At "Communication setting" select correct serial port number for the printer:
"COM14". Baudrate "250000".
Sometimes "AUTO" setting does not find the correct port and baudrate for the printer.

What is the filament diameter for MAKEiT Printer?
MAKEiT printer works with 1.75mm, with tolerance of +- 0.05mm.

How to replace guiding tube?
We can manually pull out the guiding tube by pressing down the quick fit connectors on top of the extruder module, and insert with a new guide tube.

How far does guiding tube be inserted into the extruder module?
when we insert the guiding tube into press fit on top of the extruder module, the insertion needs to go deep until it reach to the very end, about 45mm (1.75") long. If this length is NOT reached, the guiding tube is not properly installed, which will cause filament extrusion problem, such as no filament coming out; false impression of nozzle clogging, etc.

How come my part is cracking?
Increase nozzle temp by 5c increments
If it's large part, increase speed, increase layer height
Increase wall thickness, increase infil amount, and/or flow rate
Large parts built by ABS tend to crack, try to use PLA or PET, or PC-ABS (polycarbornate) for durable parts printing.

Why my print's corner is lifting up from hot bed?
Not enough heat in on hot bed: increase bed temp tuning menu while printing is going on, or reset bed temp for the next print.
slower speed if printing speed is 30-40. Usually we print at 20.

How come filament is not extruded?
Couple situations may be:
Filament is not in the heated chamber, guiding tube is not placed properly.
Filament is in extruder, but nozzle is too close to build plate
Nozzle is clogged
Filament is not installed

When to change nozzle height?
Typically we see two situations as the following:

When to change nozzle height?
Typically we see two situations as the following:
First layer of print does NOT stick to build plate. This indicates nozzles are too high from the plate, we need to bring it down.
The first layer printed is too thin. This means nozzles are too close to the build plate, we need to raise the nozzle height a bit.
We can follow nozzle height adjustment procedures in chapter 3.3 Printer Calibration, and chapter 4.3 During Printing.

What is a good guideline for printing small parts?
Slower printing speed
Higher bed temp
Faster fan speed

What is a good guideline for printing large parts?
Less infill
Faster printing speed
>0.2mm layer height

How to take "duplication setting" off from Cura if I want to switch from duplication to single print?
Under "Start/End GCode" window, highlight "Start GCode", add ";" before "M823" to de-activate duplication function.
While in SD card printing mode, as long as your file is prepared and saved without duplication gcode, then you will print single object.

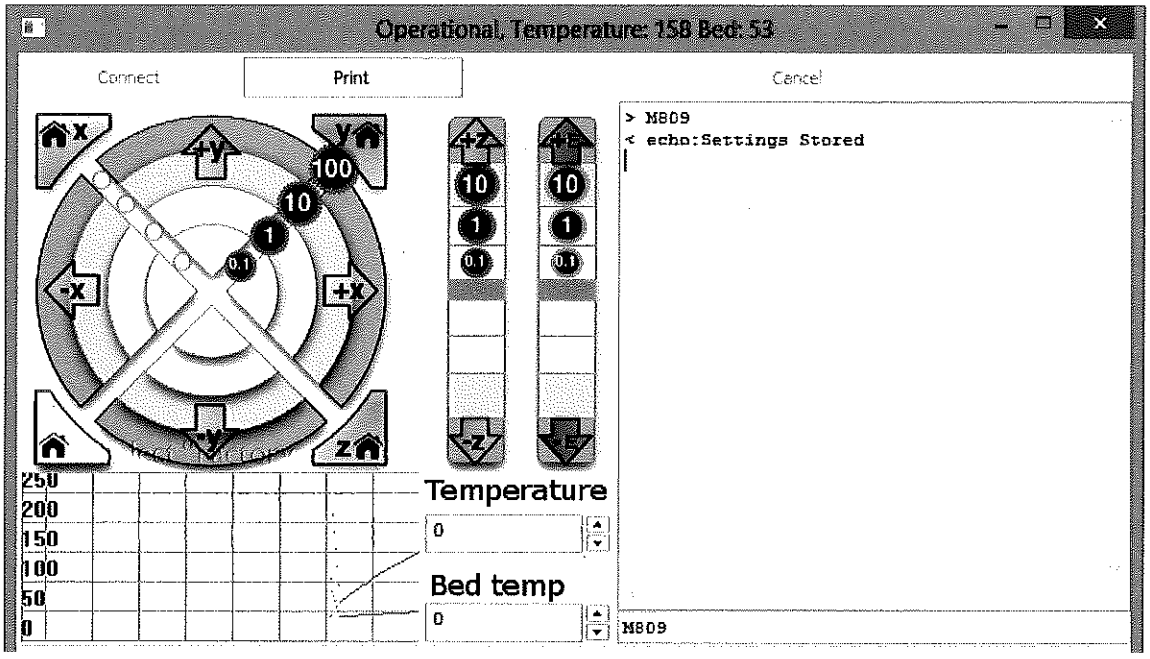
8 SUPPORT AND UPGRADE

On line support:
Trouble shooting and printing tips will soon be available online via a series of short video clips at www.makeit-3d.com. MAKEiT user group is another helpful place to find support.
Live support is always available:
By phone: 626 470 7938
Email: support@makeit-3d.com

8.1 FIRMWARE UPDATE

MAKEiT, Inc. will keep users notified and updated once new firmware release is available.
To update we can follow these steps listed below:
1. Download firmware to known location into your PC.
2. Turn power off from the printer.
3. Start Cura and goto "Machine" menu and select "Install custom firmware...".
(Ignore warnings)
4. Keep printer power switch in off position (down) manually approximately 10 seconds and after that switch printer on.
5. Select downloaded firmware (MAKEiT.cpp.hex).
6. Cura starts to send code automatically to the printer and indicates when it's ready.

After firmware update, it is good to save new settings in the printer's menu:
Select "Maintenance"
If we don't see "maintenance" command under "Setup" menu, we can add Gcode "M809" on the Cura's "connect/print" window.



TERMS OF SERVICE

9

9.1 LIMITED WARRANTY

Six month product and labor warranty is given to any original purchaser from [illegible] date, under normal use and service, against defective workmanship and materials. All original parts installed and replaced by MAKEiT Inc. or its' service provider will be warranted. Any after market modification will not be warranted. Original invoice must be presented at the time of warranty service.

MAKEiT, Inc. is not responsible for loss of work caused by product that requires service. This warranty will be void if the product has been damaged by accident, or unreasonable use, neglect, abuse, immersion in water, improper installation, improper service, or other causes not arising from defective workmanship and materials. The warranty and remedies set forth are exclusive and in lieu of all others, oral or written, expressed or implied. No reseller, agent or employee is authorized to make any modifications or extensions to this warranty without the prior written consent of MAKEiT, Inc.

Extended warranty can be discussed to meet specific needs.

9.2 RETURN POLICY

Any un-opened package can be returned within 30 days of purchase with a valid proof of purchase. A refund minus 10% re-stocking fee will be returned to customer's account. Please contact MAKEiT, Inc. first to get a RMA number before sending the return.

this policy also applies to any printers delivered and set up by MAKEiT personnel, but does not cover those printers previously placed in customer's premises for 30 day trial use. After 30 day trial period, all purchases are final.

9.3 LIMITED LIABILITY

MAKEiT, Inc. makes no other warranty, either express or implied, including, but not limited to, implied warranties of merchantability, fitness for a particular purpose, or conformity to any representation or description, with respect to its printers other than as set forth herein. MAKEiT, Inc. makes no warranty or representation, either express or implied, with respect to any other manufacturer's product or documentation, its quality, performance, merchantability, fitness for a particular purpose, or conformity to any representation or description.

MAKEiT, Inc. has limited liability that shall not exceed the purchase price less shipping and taxes. MAKEiT, Inc. shall not be liable for any direct or indirect loss, cost, expense, inconvenience or damage that may be associated with its product usage.

10

ACKNOWLEDGEMENTS

MAKEiT Team is most grateful to everyone at the workshop, the graduate student studio, especially Mr. David Cawley (Director of Rapid Prototyping and Model Shops), Mr. Bruce Dominguez,(manager, Rapid Prototyping) Mr. Tim Huntzinger (Designer and Professor of Graduate Transportation Design), Geoff Wardle (Dept. Chair, Transportation) and students at Art Center, College of Design Pasadena for countless hours of using our printers, providing us valuable feedbacks.

We also want to extend our appreciation to Mr. Powert from Carver Elementary School; Miss Renee Yang and Mr. Craig Christesen from Arcadia Creative World. They have been testing our beta printers and bringing forth great inputs and supports.