WWW.KORA.CO.UK

SPACE TO CREATE

MIDI

QUICK START GUIDE

Safety & Components

STEP 1 | Unpacking & Setup

STEP 2 | Loading & Unloading Filament

STEP 3 | Preparation & Printing

STEP 4 | Switching the Printer Off

STEP 5 | Calibrate Print Bed

SAFETY & COMPONENTS

IMPORTANT SAFETY INFORMATION

[READ THIS FIRST]

All printer beds have completed a successful test print prior to delivery or collection.

DO NOT leave the printer unsupervised while switched on or running.

DO NOT place hands inside the printer while it is switched on – even if not printing.

DO NOT use the printer on an unstable surface, table or bench.

DO NOT operate the printer if there is any mechanical or wiring damage.

Always inspect the printer and cables before use.

DO NOT switch the printer off until it has fully cooled down and there are no warning

 lights visible (see Step 4 on Page 7).

DO NOT allow any cables to become a trip hazard.

Take care when lifting and moving the printer – the printer is heavy.

Make sure the mains connection provided is safe – if in doubt DO NOT use.

If the printer is connected to a mains extension lead make sure the extension

lead is fully unwound.

IN THE BOX

Bed adhesive
Power lead
Power supply
Drill bit (unblocks nozzle)
Hot end cleaner spike
Custom 'Filament Ooze from Nozzle' Tweezers
Usb lead
Print Bed Scraper
Allen keys
SD Card
Spool holder
Test print (spool holder and feet)

STEP 1

UNPACKING & SET-UP

Remove the top internal packaging. Firmly hold the left and right sides of the

printer and carefully lift the printer out of the box.

TAKE CARE WHEN LIFTING AND MOVING. THE PRINTER IS HEAVY!

CAUTION

Place the printer on a bench or table. Make sure the surface,
table or bench is level and stable.
Unpack the accessories box that is placed under the
Place the SD card into the Printer SD card slot.
Connect the Mains Lead.
The Printer digital display will light up

STEP 2

LOADING & UNLOADING FILAMENT

If the Filament is new, remove all packaging.

Fit the Printer Filament Spool Holder to the printer. Insert the filament spool holder through the filament reel. Fit the filament spool holder with the filament attached onto the printer, place through the top hole on bracket on the right back of printer secure with wing nut provided.Filament needs to be loaded from the back of the reel directly up to the feeder.

When loading Filament make sure the end of the Filament is cleanly cut straightened at the end to help loading.

First heat up the extruder, to do this on the screen go onto quick settings and select heat PLA/ABS (depending which filament you are using) press toggle.
Using' back' on the screen go back to menu screen and this will show you the temperature of the extruder, when temperature is reached remove any filament that is already loaded by squeezing/pinching the two blocks on side of printer compressing the spring and pulling filament out. (if changing filament).
To load new filament squeeze/pinch the two blocks on side of printer compressing the spring ensuring that the filament has no bends push up through the bottom of the feeder ensuring that it goes into the plastic tube, keep pushing until you see the filament coming out of the extruder, push until you have aprox 6 inches out of the extruder or until the colour you are now using runs clean. Once filament is loaded back go to quick settings scroll down to cool down, this will bring extruder temperature down, do not turn printer off until the red light on the hot end has gone off. The Hot End is the Printer Head Nozzle. The red light is displayed a few centimetres above the Hot End on top of the circuit board."

STEP 3

CALIBRATE PRINT BED

It cannot be stressed enough levelling the printers bed correctly is essential to achieve good adhesion between the printer bed and the first layer of all your 3D prints. Only **VERY LIGHT** contact is required between the tip of the brass nozzle and the printer bed, but **NO** "gap" should be present.

Before completing bed levelling below **make sure** there is no filament on the tip of the brass nozzle or anywhere on the printer bed. To remove filament on the nozzle it may be necessary to heat it up

using the Preheat option found in >Main menu >Quick settings >Preheat PLA or ABS (see fig 4 below). When heated, remove any filament using metal tweezers, a small pair of long nose pliers or a piece of thick card. CAUTION – IF HEATED, NOZZLE BECOMES VERY HOT – AVOID SKIN CONTACT. Filament or debris on the printer bed can be removed using the scraper provided.

Levelling the printer bed when the nozzle is at 'printing' temperature is ideal – this allows any filament present to be squeezed out of the way when molten – allowing accurate judgement of light contact between the nozzle and printer bed.

There are several ways to ensure that the bed is level and ready to print.

A. From SD card:

1. Insert SD card (if already inserted remove and re-insert). On the file list shown on the Lcd screen scroll down and select the file named 'midi levelling.gco' (see fig 3 below). The printer will home all axes, then move the nozzle to far right hand corner slowly raising the bed towards the nozzle.
2. When the bed stops rising check that the nozzle is **lightly** touching the bed surface. This is adjustable using the thumb screw under the far right hand corner of the bed. You have 10 seconds after the bed has stopped rising to make this adjustment correctly. After 10 seconds the bed will slightly lower and the nozzle will move to the far left hand corner to repeat the process in 2 above.
3. After 10 seconds the bed will slightly lower and the nozzle will move to the near left hand corner to repeat the process in 2 above.
4. After 10 seconds the bed will slightly lower and the nozzle will move to the near right hand corner to repeat the process from 2 above.
5. The nozzle will now go the centre for 5 seconds to do a test this shouldn't need any adjustment.
6. The screen will then display 'One quick check!' and will repeat everything from 2 above but this time only pausing at each corner for 5 seconds.
7. You can repeat the process, start again from 1 above.

B. Manual Method:

1. Removing the power supply plug to switch off the printer, replace plug to switch the printer back on and wait for 'boot up' sequence to complete.
2. Press jog wheel once and scroll down to "Position" (see fig 1 below), scroll down to 'Home Z' (see fig 2 below) and select by pressing the jog wheel, the bed will raise towards the nozzle.
3. When it reaches the top using the nozzle mechanism support bars manually move the nozzle to all four corners of the printer bed – at each corner use the thumb screw below that particular corner of the printer bed plate to adjust the printer bed up and down so that it **lightly** touches the brass tip of the nozzle.

4. When complete re check all four corners again.
5. You have 6 minutes to complete the above process after selecting 'Home Z' in 2 above, after this time the Z motor will time-out – if this happens repeat from 2 above.

C. From Computer:

1. In Repetier-Host, click on 'Load' on the main menu bar, navigate to the location where you have stored the file named 'midi levelling.gco' and select it.
2. Now click 'Start Print' on the main menu bar. The printer will home all axes, then move the nozzle to far right hand corner slowly raising the bed towards the nozzle.
3. When the bed stops rising check that the nozzle is **lightly** touching the bed surface. This is adjustable using the thumb screw under the far right hand corner of the bed. You have 10 seconds after the bed has stopped rising to make this adjustment correctly. After 10 seconds the bed will slightly lower and the nozzle will move to the far left hand corner to repeat the process in 2 above.
4. After 10 seconds the bed will slightly lower and the nozzle will move to the near left hand corner to repeat the process in 2 above.
5. After 10 seconds the bed will slightly lower and the nozzle will move to the near right hand corner to repeat the process from 2 above.
6. The nozzle will now go the centre for 5 seconds to do a test this shouldn't need any adjustment.
7. The screen will then display 'One quick check!' and will repeat everything from 2 above, but this time only pausing at each corner for 5 seconds.
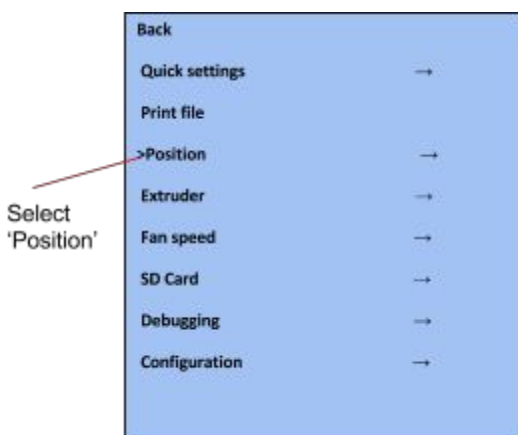8. You can repeat the process, start again from 2 above.
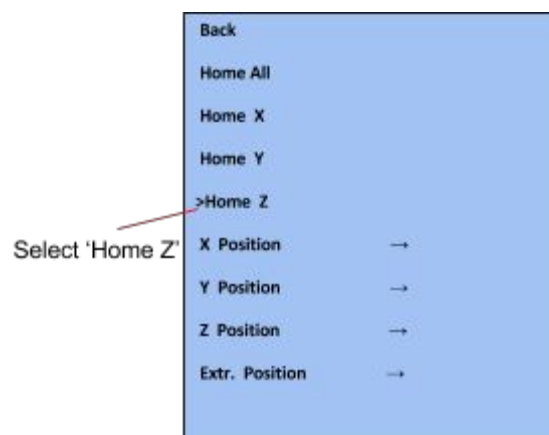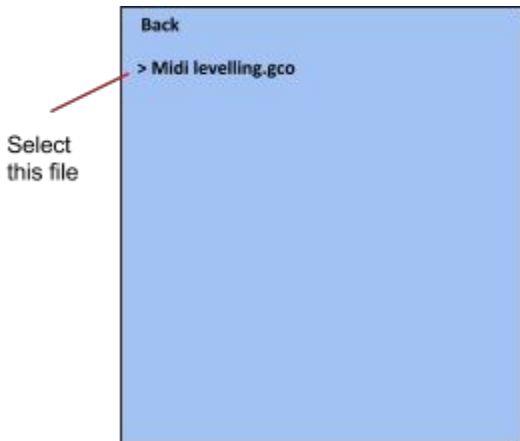
Fig 1

Fig 2

Back
Quick settings   →
Print file
>Position   →
Extruder   →
Fan speed   →
SD Card   →
Debugging   →
Configuration   →

Select 'Position'

Back
Home All
Home X
Home Y
>Home Z
X Position   →
Y Position   →
Z Position   →
Extr. Position   →

Select 'Home Z'

Fig 3

Fig 4

Back

> Midi levelling.gco

Select
this file

>Back

Home all

Z Babystep, : 0.00mm

Speed Mul . : 100%

Flow Mul .   : 100%

Lights : On

Change filament

Select filament
type

Preheat PLA

Preheat ABS

Set to origin

Disable stepper

ATX power on/off

Step 4

Preparation and printing

The Bed should be clean and free from any old print matter. Use the scraper to remove any filament. Use Kora Diamafix for heated beds or Kora sprayfix for printers with a non heated bed. The adhesive will keep the first layer of your print in position on the bed.The print area should be lightly sprayed for every print. For the first print it may need several layers of spray.

Use the Jog Dial to select an item to print. On the on screen Menu select: 'Print file' this will give you all the prints on your SD card, or 'SD Card' this will give you 'print file'.
Scroll to the print you want and press jog wheel..
The print will not start immediately!
The Hot End will move and begin to heat up showing a red warning light.

CAUTION   DO NOT TOUCH AS THESE AREAS ARE HOT.

When the Heated Bed and Hot End reach the required temperature the print will

start. The Digital Display will show the status of the print. Always monitor the first layer of any print to ensure good adhesion to the bed. When the print has finished the Bed will lower and the Hot End will move to the corner.

CAUTION ALLOW THE PRINTER TO COOL

STEP 5

SWITCHING PRINTER OFF

Switch the printer off by unplugging the power cable from the printer (or by turning it off at the plug socket).

CAUTION   BEFORE PRINTER IS SWITCHED OFF, IT MUST BE ALLOWED TO COOL DOWN

CAUTION   NEVER SWITCH THE MACHINE OFF IF THE RED WARNING LIGHT ON THE TOP OF THE HOT END IS ILLUMINATED

Repetier™ firmware

Disclaimer

The information provided into this booklet is provided "as-is" and proved to be exact as supplied with 3ntr A4 printers.

Up to date with Repetier Firmware version 0.83

Values depicted and other settings are just for explanations: your setup may differ (polymer make, color, precision needed) .

In no case Jdeal-Form srl will be liable for any money loss deriving from the reading of this booklet All © and ™ are property of respective owners.

Get to know more about Repetier™ software, visit www.repetier.com !

If you use Repetier firmware in a professional setting, please donate:
http://www.repetier.com/donate-or-support/

Last updated: 24th June 2015

Main LCD screens

At startup, the display shows following data :

```
E1:  23.9/    0°C→    0%
Z: 00.00 mm
Buffer: 0
Idle
```
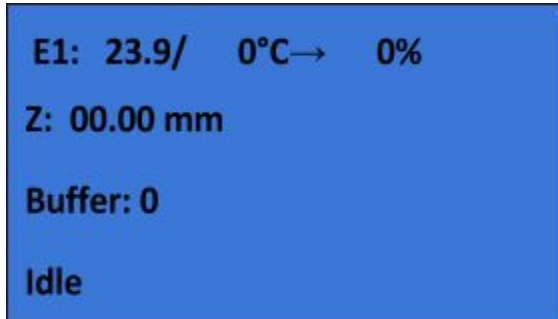
The first line shows current extruder temperature (23.9°) and target temperature (0°), % of power currently being supplied to the heater cartridge. (0%)

The second line shows current Z position (00.00).
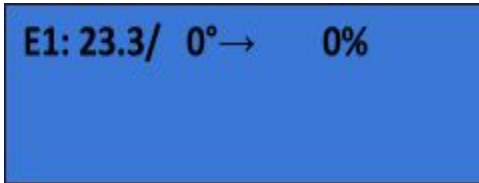
The third line shows current speed multiplier (100) as percentage of programmed speed*and buffer capacity (0) expressed as moves buffered.

The fourth line tells the current machine status (Idle).

Moving the jog wheel, you can display another page, showing the current extruder position:

```
X: 00.00 mm
Y: 00.00 mm
Z: 00.00 mm
Idle
```

Further scrolling with the jog wheel you will get to heater value/setting/power positions:
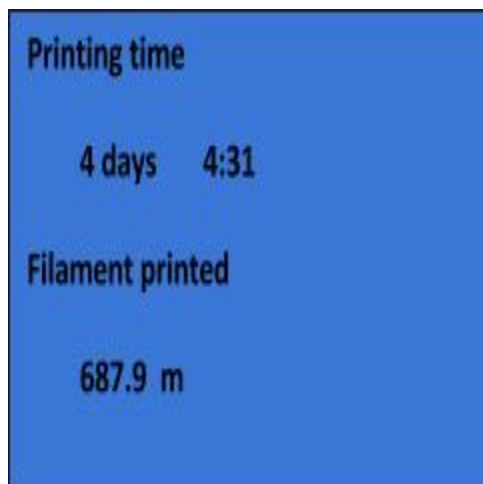
E1: 23.3/ 0°→   0%

Idle

For the extruder you can read the current temperature, the target temperature and the power currently being supplied to the heater cartridge.

Further scrolling with the jog wheel you will get to the printing information screen below (since the last firmware update)
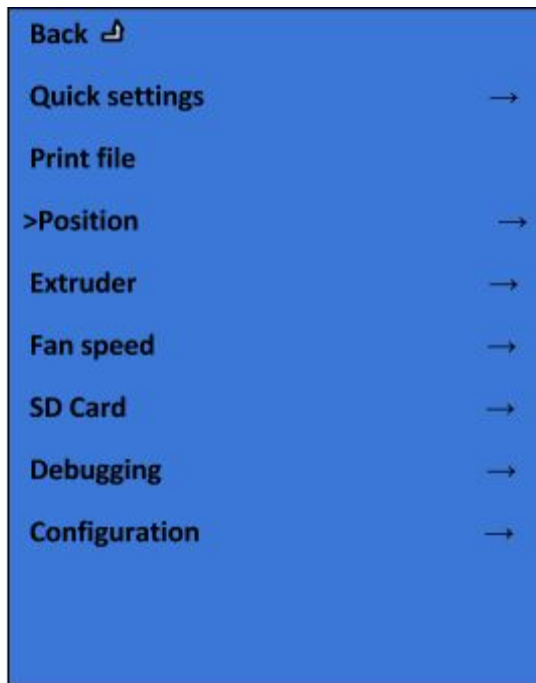
Printing time

4 days   4:31

Filament printed

687.9 m

Main LCD menu

Pushing the back button in any of the main LCD screens you are presented with the main menu structure:

```
Back  ⏎

Quick settings          →

Print file

>Position               →

Extruder                →

Fan speed               →

SD Card                 →

Debugging               →

Configuration           →
```

Pushing the black button again will get you back to the main LCD screens.

Scrolling the ">" symbol on requested item and pressing the black button will activate the requested item menu.

1. **Main->Quick settings**

```
Back              ⏎
Home all
Z Babystep,  : 0.00mm
Speed Mul . : 100%
Flow Mul .    : 100%
Lights  : On
Change filament
Preheat PLA
Preheat ABS
Cool down
Set to origin
Disable stepper
ATX power on/off
```

Home all

Will move all axes to homing positions.

Warning: be sure start this command when printing area is empty !

Z Babystep

Allows you to adjust the height of the nozzle " " live while extruding the first layer of filament, in case the first layer appears not to be adhering to the bed, or is extruding too close to the bed causing the extruder motor to " " click continuously.

Speed Mul.

When printing, will let you override programmed feed rates.

Warning: you may actually cause filament jams if you set speed too high for extruder to bear.

Flow mul.

When printing, will let you override programmed filament feed. It is useful when researching correct flow rates.

Warning: you may actually cause filament jams if you set flow rate too high for extruder to bear.

Lights on.

Lights on/off option

Change filament

Heats the nozzle then removes the filament from the filament tube automatically allowing you to replace the filament and wind it up the tube using the jog wheel until melted filament appears out of the nozzle.

Preheat PLA

Will set extruder to 220° C (default values may be changed when reflashing firmware).

Preheat ABS

Will set extruder to 240°C (default values may be changed when reflashing firmware).

Cooldown

Bring extruder to cool down temperature

Disable stepper

Power off all axes, letting user move them by hand.

ATX power on/off

Not applicable for Kora printers

2. **Main->Print file**

A lists of all prints stored on machine will appear here.

3. **Main->Position**
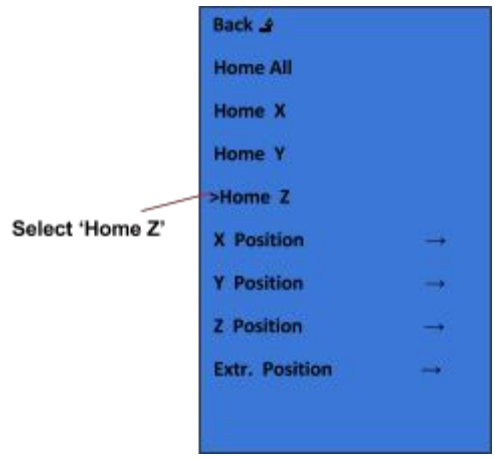
Select 'Home Z'

Home all

Will move all axes to homing positions.

Warning: be sure start this command when printing area is empty !

Home X / Home Y / Home Z

Will move selected axis to the home position.

X / Y / Z Position

This option makes precise movements: each turn of the jogwheel equals 1mm approx movement

Extr. Position

This option lets you jog the filament feed: be sure to use it when extruder is heated at correct setting ! Also, excessive retraction may cause filament jams with some filaments.

4. **Main->Extruder**

Back ↵

Temp . 1 :   0°C

Turn Extr . 1 Off

Extr . Position   →

Set origin
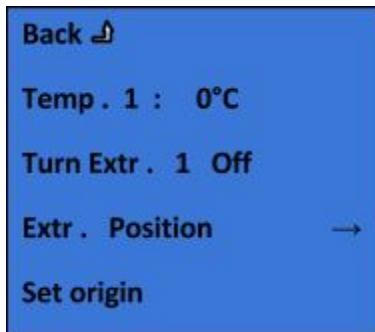
Temp. 1

This option lets you override extruder temperature settings

Extruder 1 OFF

To turn off extruder (temperature is immediately set to 0)

Extr. Position

This option lets you jog the filament feed: be sure to use it when extruder is heated at correct setting!

Excessive retraction may cause filament jams with some polymers.

Set origin

Will reset the extruder feed position counter.

5. **Main->Fan speed**

Back ⏎

Fan speed: 0%

Set Fan 25%

Set Fan 50%

Set Fan 75%

□

Set Fan Full
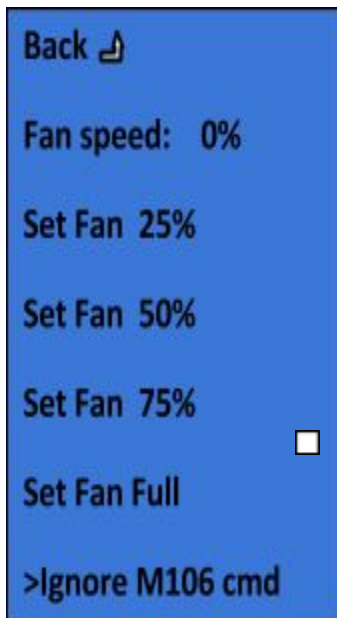
>Ignore M106 cmd

Fan Speed

Controls fan speed

Set fan 25%-50%-75%-Full

Sets the print area fan speed.

Ignore M106 cmd

Not applicable on Kora

6. **Main->SD Card**

> Back↵
>
> Print File
>
> Delete File
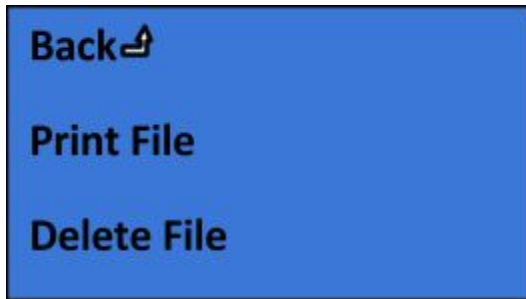
Print file

Selecting this option, you will be shown the list of files available on the printer SD card: you can browse and select the file name to print (or hit Back to abort the selection process).

 The selected file name will be run immediately.

Pause Print

To temporary stop the printing process.

This option will only appear when a print is running

Continue Print

To resume the paused printing process

Delete file

To permanently remove files from the memory card

7. Main->Debugging

```
Back ↵
Echo    :off
Info    :on
Errors  :on
Dry run:off>
```

Echo

Will enable sending to host (your PC connected to the printer) feedback about the command received

Usually this option is left off (to avoid wasting communication time)

Info

Will send to host (your PC connected to the printer) diagnostic messages.

Error

Will send to host (your PC connected to the printer) error messages.

Dry run

Will ignore any extrusion command: useful to test program movements without actually wasting polymer.

8. Main->Configuration

Back ⏎

General →

Acceleration →

Feedrate →

Extruder →

Store to EEPROM

Load f. EEPROM

Level delta →
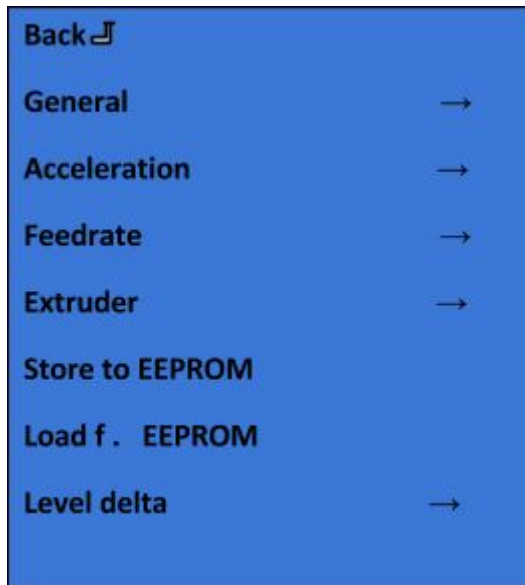
Be careful with this menu: here you can alter values that could prevent you from being able to use your printer!

Moral: be sure to know what you are doing: write down previous values when you attempt new ones!

It is wise to always write down the actual values you are going to change, to let you regain control if something goes wrong!

Also, changing parameters while printer is running a job may bring unpredictable results.

Main->Configuration->General

```
Back ↵

Baudrate:250000

Stepper Inactive          →

Max. Inactive             →

Autolevel :  off
```
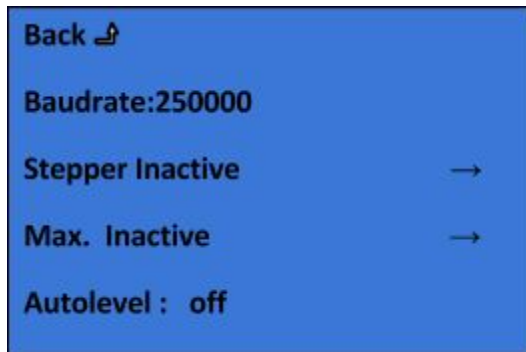
Baudrate

Sets the speed used to talk with the PC: this value must match the one set on your PC host software!

Stepper inactive

Sets the stepper motors inactivity timer: after chosen time, the motors will be powered off and axes position won't be held anymore. Setting to zero disables it (motor will be always powered)

Default is two minutes (120 sec) you set this value to avoid unneeded waste of power after finishing an unattended print.

Max inactive

Sets the system inactivity timer: after chosen time the extruders and heated bed are turned off.

Default is zero (disabled). The heating strategy is set with G code (printing files).

Main->Configuration->Acceleration

```
Back ↵

Print    X:  2000

Print    Y:  2000

Print    Z:   100

Move  X :  3000

Move  Y :  3000

Move  Z :   100

Jerk      :  20.0

>Z-Jerk  :   0.3
```
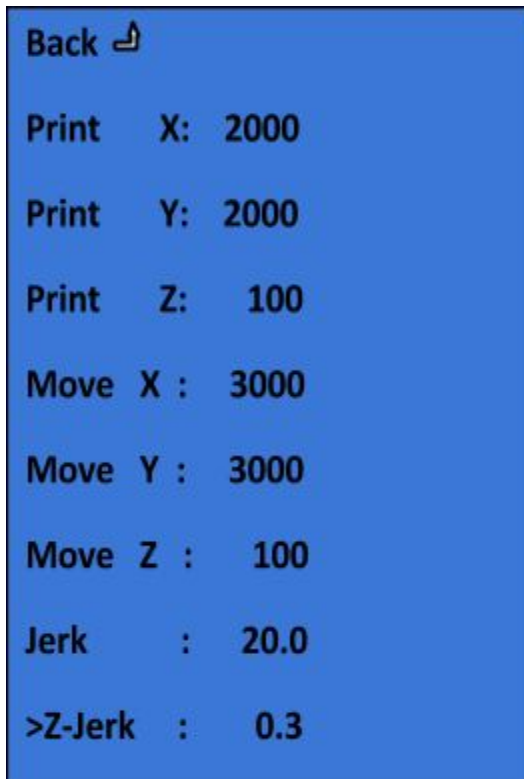
What is acceleration?

It is the times it takes to change from zero to the desired speed, and it is expressed as mm/sec squared.

With higher acceleration rates:

    Machine runs faster
    Corners are printed sharper
    More noise
    Drives and controller electronics run hotter
    Possibility of losing steps (positional accuracy) is increased
    Machine hardware is more prone to vibration (and the possibility of loosened screws is increased)

With lower acceleration rates:

    Machine runs slower
    Corners are rounder
    Quieter printing
    More precision, as machine runs cooler and there are less chances of missing steps

Print X/Y/Z

Changes the acceleration rate used when printing (units are mm/sec*2) Too low values won't let your machine reach the maximum speed.

Too high values may ask for acceleration too fast for the machine hardware (loosing steps or overheating)

Move X/Y/Z

Changes the acceleration rate used when moving (units are mm/sec*2). Too low values won't let your machine reach the maximum speed.

Too high values may ask for acceleration too fast for the machine hardware (loosing steps or overheating)

Jerk

It is a value that sets the "joining speed" of consecutive segments.

Higher values give you faster speeds and nicer corners (but higher noise and possibly lost steps)

Lower values give longer prints and smoother movements (quieter and definitely more precise prints)

Z-Jerk

It accounts for Z movement inversion.

Usually is best left at less-than-one values.

Main->Configuration->Feedrate

```
Back ⤵

Max  X:     300

Max  Y:     300

Max  Z:      20

Home   X:      40

Home   Y:      40

Home    Z:    20
```
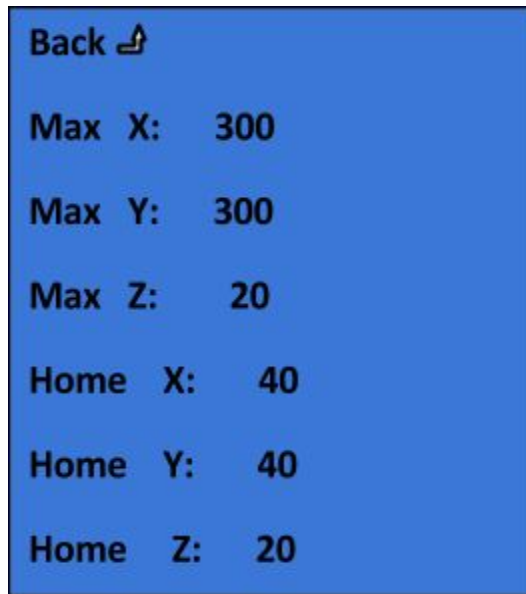
What is feedrate?

It is the axis speed, expressed as mm/sec.

Max X/Y/Z

The maximum speed your system may tolerate: any faster rate from printing instruction will be limited at those values.

If set too high you may experience lost steps (loose precision!)

Home X/Y/Z

The speed you want to use when seeking the home limit switches.

Usually, slower speed allow for better precision, especially if the electronics are somewhat slow-performing.

Main->Configuration->Extruder

```
Back ⮌

Steps/MM: 518.0

Start  FR:        20

Max FR :    50

Accel :  5000

Stab . Time :    1

Wait   Units :    0  mm

Wait   Temp :    200°C

Control : PID

PID   P :      13.46

PID   I :          3.59

PID  D  :      12.62

Drive   Min :    40

Drive   Max :   200

Pid   Max :   255
```

Steps/MM

How many steps are needed to feed one mm of filament

Start/FR

Minimum feedrate (mm/minute) for the extruder motor

Max/FR

Maximum feed rate (mm/minute) for the extruder motor

Acceleration

Acceleration (mm/sec2) for the extruder motor

Stab.Time

Time to wait (in seconds) when temperature is reached, to let system settle, before starting the

Wait units

How much filament must be retracted when heating up.

Printing process

Wait temp

Automatic retraction (when waiting) will be performed after reaching this temperature

Control: PID

This option lets you choose between PID and BANGBANG heating strategy.

The PID lets you precisely tune the heating strategy, but can be overwhelming for novices and usually cannot be used with traditional relay devices (the PID process may ask for tens or hundred on-off cycles per minute, easily burning any relay... use this strategy ONLY if you have SCR or solid-state relays)

The BANGBANG is simply ON until temperature is below the set limit, and OFF when temperature is reached. Usually this strategy is recommended for heated beds.

PID P/I/D

The parameter that set the rules for the PID heater.

Usually all you need to do is sending the "M303 Px Syyy" to the printer (where x stands for the extruder # and yyy the desired working temperature), then copying the found values into those fields.

Why one should perform the PID auto tuning (aka "M303....")?

Usually if you always worked with PLA and suddenly decide to move to a much higher melting point polymer, you may experience a bit less precise temperature readings... if you can live with this, no problem. If you want topmost precision, then you know what to do.

Drive Min

Sets the minimum I-drive value (the "I" in PID). Usually left low enough to allow for cooling phase (value range 0..255)

Drive Max

Sets the maximum I-drive value (the "I" in PID). Usually left high enough to reach desired temperature (value range 0..255)

PID Max

Set the maximum voltage you can send to your heater (0..255).

If your heater is rated at same voltage of your power supply, let the value at 255.

If your heater is rated at a fraction of your power supply (i.e.: 12V heater with a 24V power supply) you should start with values around ¼ of full setting (64) and slowly increase until you are satisfied with your PID behavior. Do note that PID should be re-tuned each time you change PID_Max.

Main->Configuration->Store to EEPROM

You choose this option when you have changed some configuration parameter and you are fully satisfied with the results: writing the value into EEPROM will make sure that next time you will turn on printer it will start with new settings.

Main->Configuration->Read f. EEPROM

Use this option to overwrite current machine setup with last saved settings.

Level delta

Not available on Kora

Don't be an obsessive printer and polluter:

–Don't print this manual, rather keep it as a PDF file on your PC / Laptop / Pad / E-reader / Smartphone / Clock / brain implant ROM / ....Conserve trees, energy and paper!

–ANY plastic is an environmental costly medium. Therefore think twice before printing yet another yoda™ that nobody will really need.

–Infill (how much dense is the inside of your models) will add cost and time to your prints. Usually 20% is enough for 80% of prints.

–Any polymer will pollute (yes, even PLA) when melted.

> Volatile compounds will develop, especially with darker shades of color.
> Don't use your printed in a small un-vented closet.
> Keep an open window when printing (especially when using higher melting points products).
> Enclose your printer in a box, to confine smoke.
> Recycle your scrapped prints and rafts!

KORA

Repetier-Host Documentation

Installation

Prerequisites

Before you start with the installation, you should check if your computer meets the requirements. Currently available computers should have no problems at all. If you have an old computer running Windows XP you may have difficulties. The Repetier-Host works on Windows XP and later and on Linux. If you have a Macintosh computer, check for the Repetier-Host Mac on this site. All you need is .NET framework 4.0 or a recent Mono installation, if you are running Linux. The only other requirement is a graphic card with OpenGL. For a good rendering performance OpenGL 1.5 or higher is needed. With lower versions you may have speed issues with live preview.

Getting the software

Go to the Repetier-Host ([download page here](#)) and download the latest version for your operating system.

Now, go to the Arduino ([download page here](#)) and download the latest version for your operating system.

Windows installation

First, go to the folder you downloaded the Arduino file into and run the

arduino-1.6.0-windows.exe file (or latest version) by right clicking on it and choose the 'RUN AS ADMINISTRATOR' option – follow the on-screen instructions to install the program

Next, The Repetier-Host windows file comes with an installer. Go to the folder you downloaded the Repetier-Host file into and run the setupRepetierHost_6.exe file (or latest version) by right clicking on it and choose the 'RUN AS ADMINISTRATOR' option – follow the on-screen instructions to install the program

 The Repetier-Host installer already contains Model slicing software Cura and Slic3r.

Configuration

Configuring your printer

The next step is to configure the Repetier-Host software so your printer can connect to your computer. If you have installed the 'Arduino' program (see page 1), the driver will already be installed.

Go to the menu "Config"->"Printer Settings" or click the button Printer Settings. You will see a window like this:
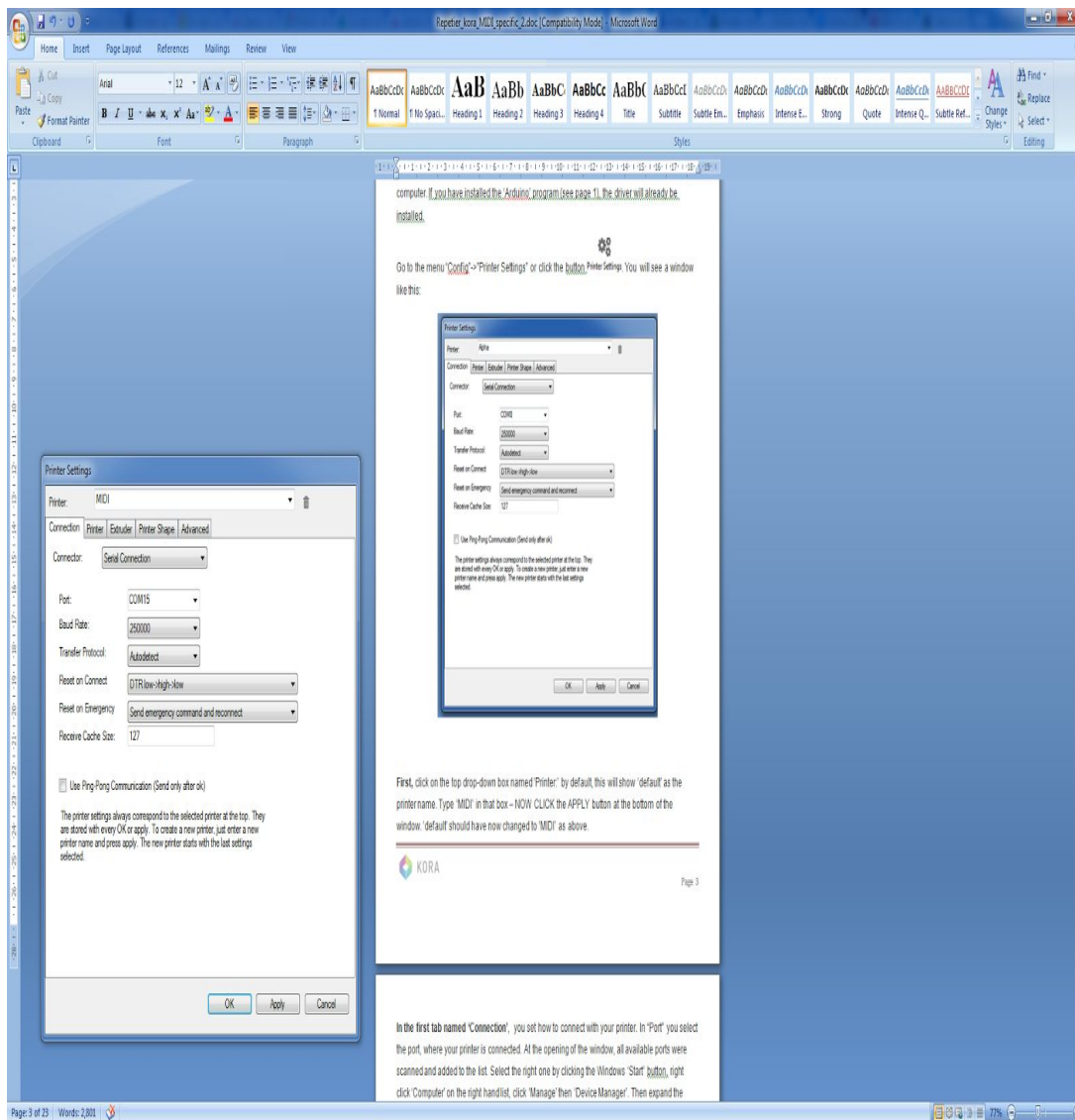

First, click on the top drop-down box named 'Printer:' by default, this will show 'default' as the printer name. Type 'MIDI' in that box – NOW CLICK the APPLY button at the bottom of the window. 'default' should have now changed to 'MIDI' as above.

In the first tab named 'Connection', you set how to connect with your printer. In "Port" you select the port, where your printer is connected. At the opening of the window, all available ports were scanned and added to the list. Select the right one by clicking the Windows 'Start' button, right click 'Computer' on the right hand list, click 'Manage' then 'Device Manager'. Then expand the PORTS (com & lpt) – with your Kora MIDI connected to a USB port on your computer, the com port number is shown as 'Arduino' or 'USB Serial Port' with the Com port number in brackets after it. Enter this number in the Port drop-down box. Then set all other boxes. NOW CLICK the APPLY button at the bottom of the window
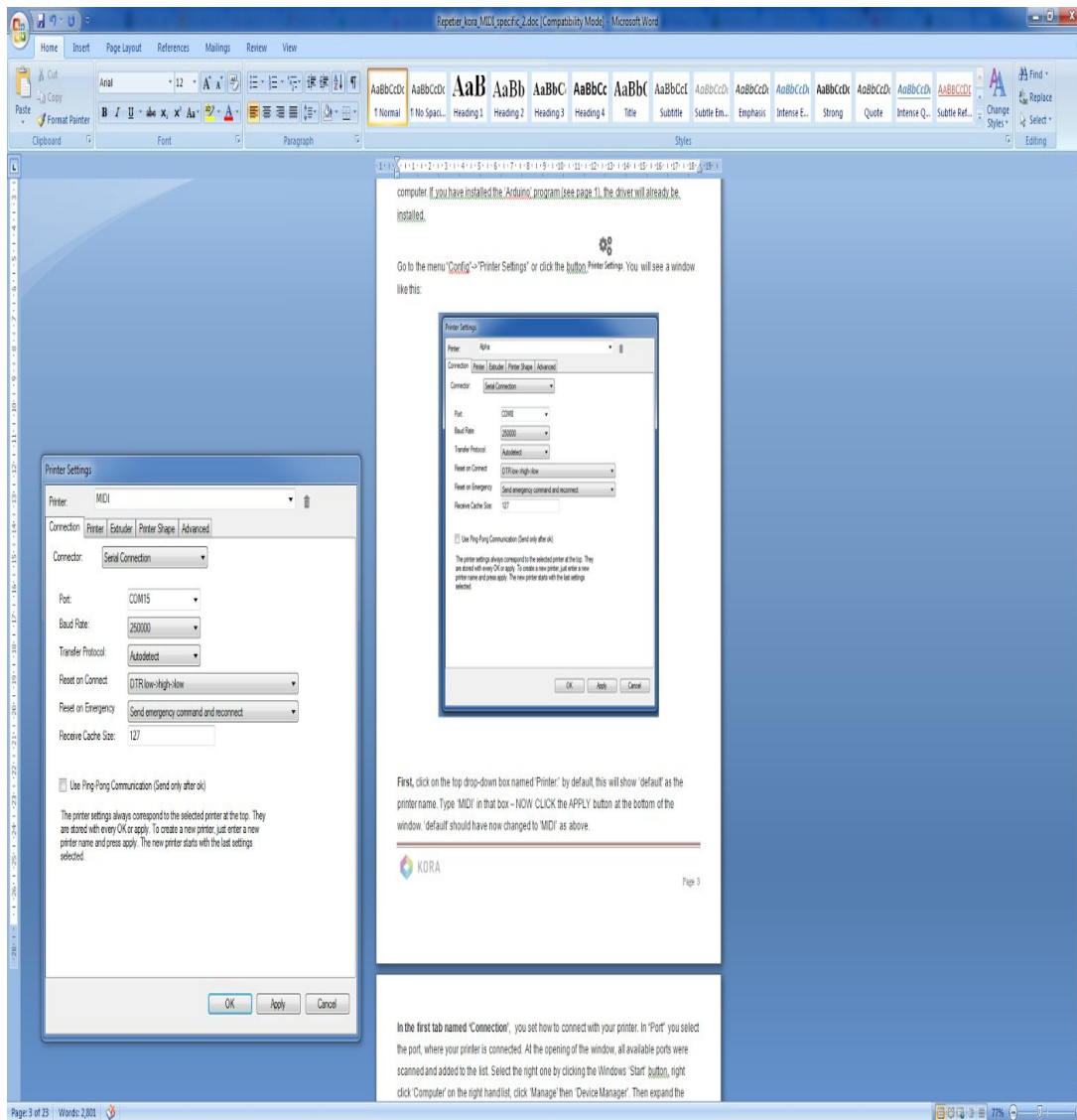
In the second tab named 'Printer', this defines important behaviour you want. Copy EXACTLY the settings shown below into the window. NOW CLICK the APPLY button at the bottom of the window
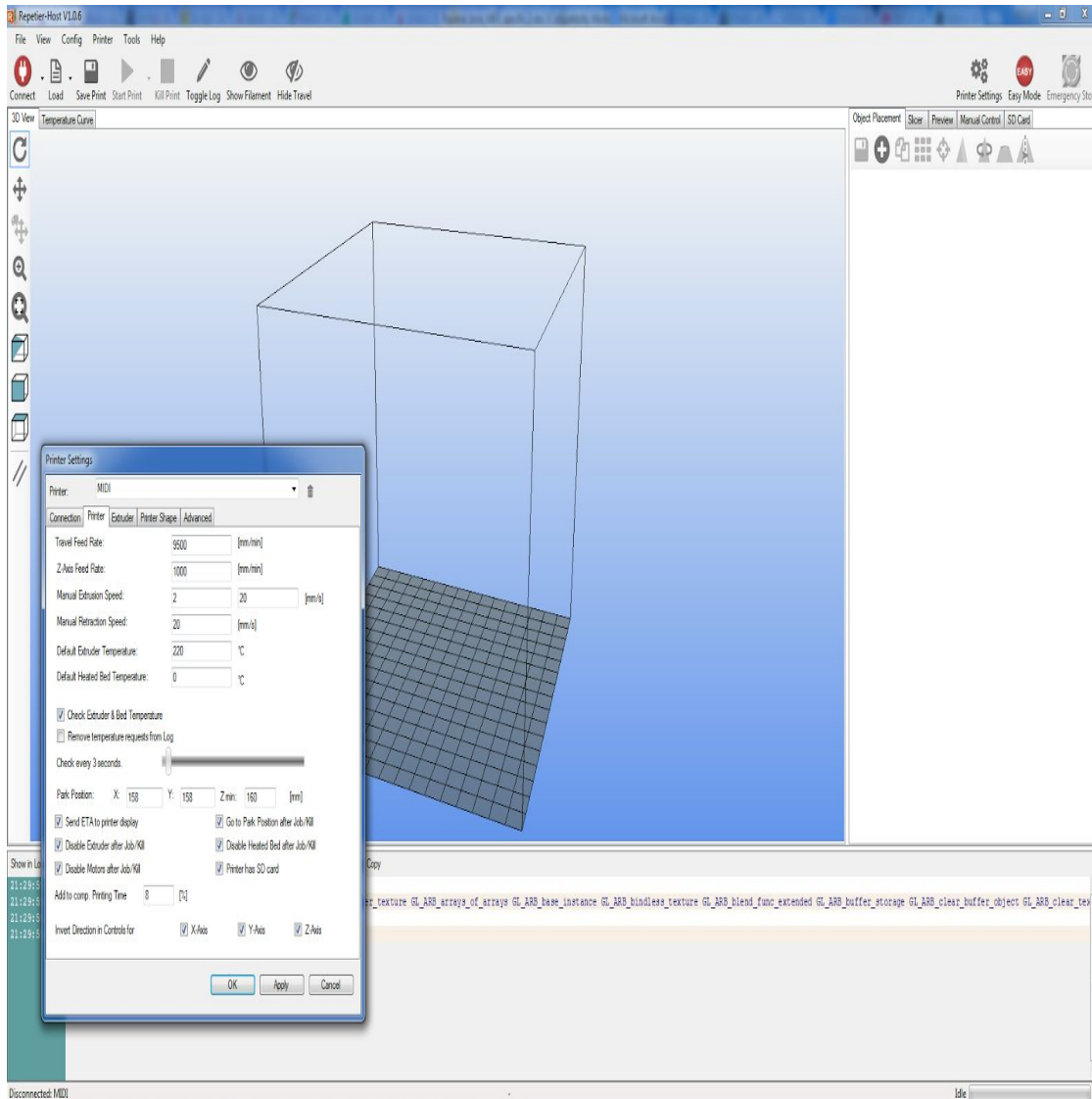
In the third tab named 'Extruder', this tab sets some important information about the Kora MIDI 'hot end' or 'extruder'. Copy EXACTLY the settings shown below into the window. NOW CLICK the APPLY button at the bottom of the window

Printer Settings

Printer: MIDI

Connection | Printer | Extruder | Printer Shape | Advanced

Number of Extruder: 1

Max. Extruder Temperature: 260

Max. Bed Temperature: 120

Max. Volume per second: 12 [mm³/s]

☐ Printer has a Mixing Extruder (one nozzle for all colors)

Extruder 1

Name:

Diameter: 0.4 [mm]   Temperature Offset: 0 [°C]

Color:

Offset X: 0   Offset Y: 0 [mm]

OK   Apply   Cancel

In the third tab named 'Extruder', this tab sets some important information about the Kora Alpha 'hot end' or 'extruder'. Copy EXACTLY the settings shown below into the window. NOW CLICK the APPLY button at the bottom of the window

KORA

Page 5

35

In the fourth tab named 'Printer Shape', this tab sets some important information about the Kora MIDI print area sizes. Copy EXACTLY the settings shown below into the window. NOW CLICK the APPLY button at the bottom of the window
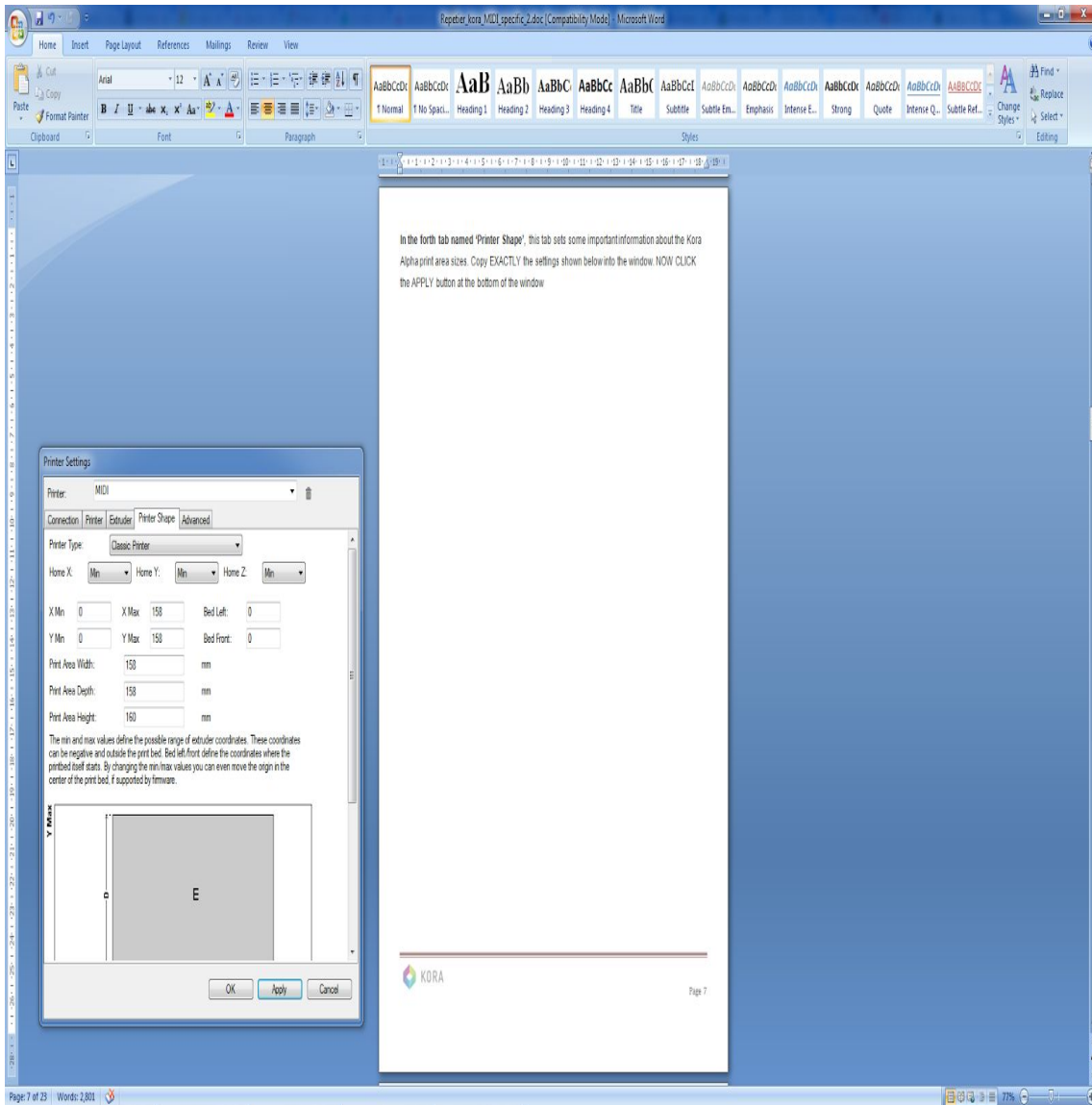
In the fifth tab named 'Advanced', this tab is for advanced users and should be left as default. Check it is showing as the window below. NOW CLICK the APPLY button at the bottom of the window
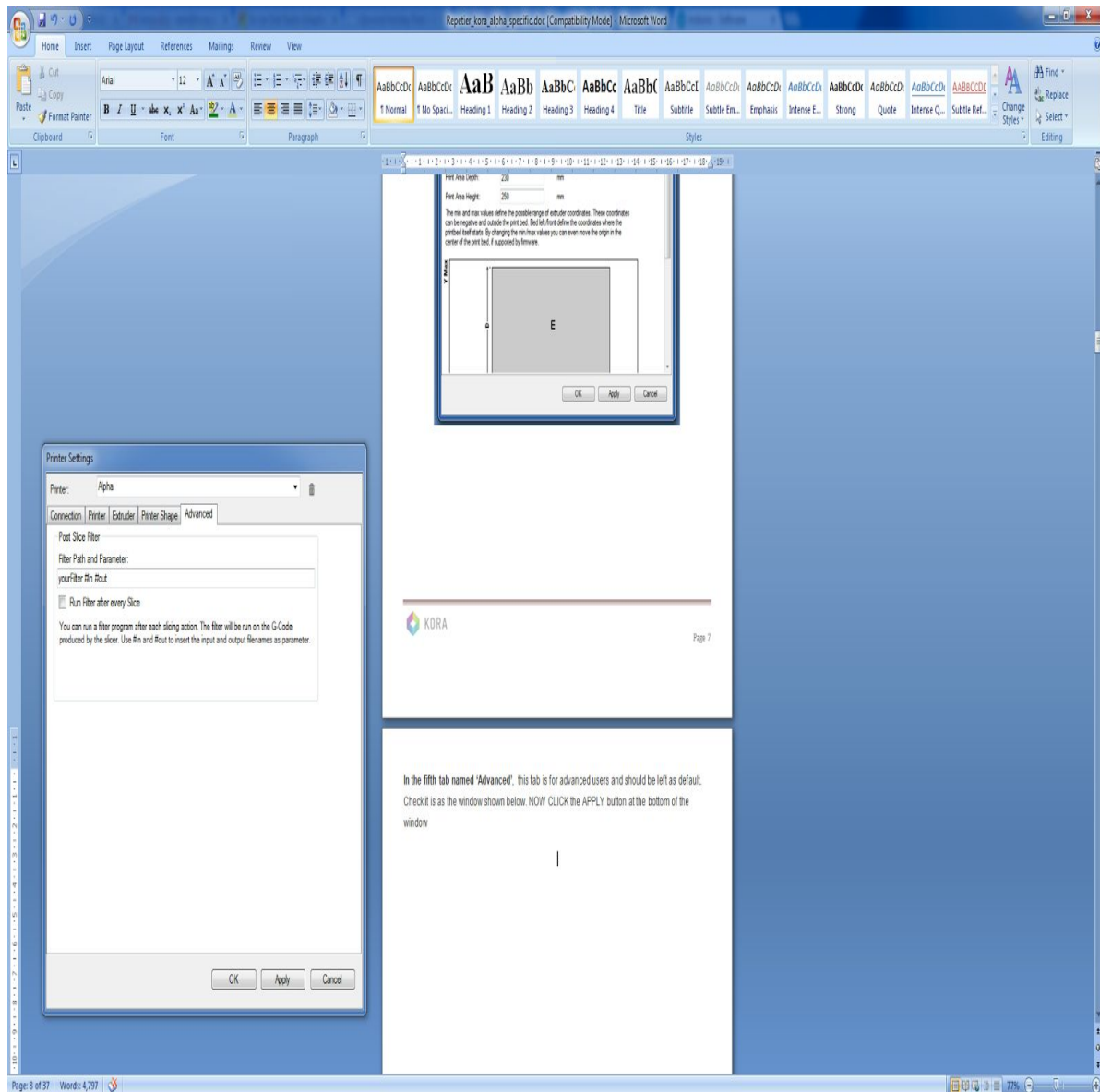
NOW CLICK the OK button at the bottom of the window.

Your Kora MIDI is now configured to function correctly with the Repetier-Host software. This information will be automatically saved and it is not necessary to repeat this process whenever the Kora MIDI is used on this computer.

Units Settings

From the 'Main Screen' on Repetier-Host, check the following settings are applied
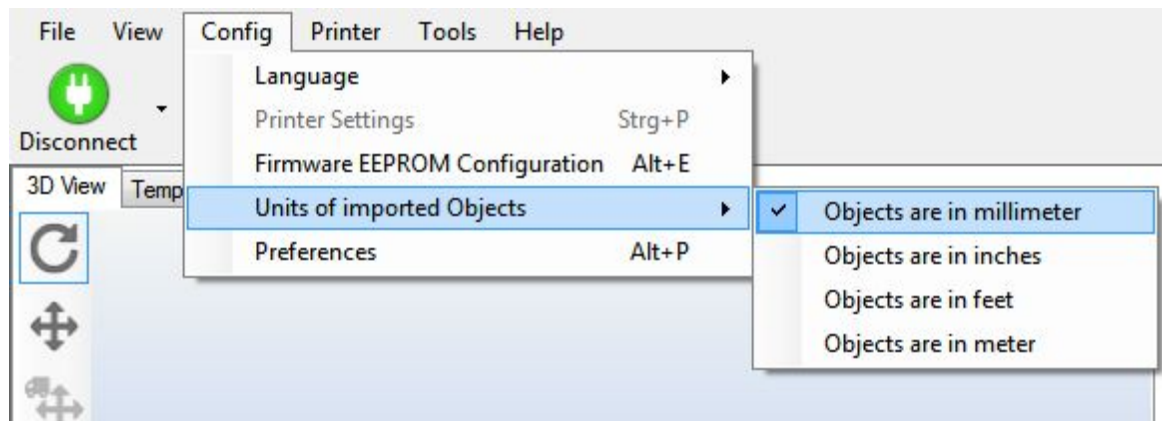
The Repetier-Host uses internally only millimetres. Unfortunately STL and OBJ format do not contain a scaling hint. Therefore you need to select the units used for the creation. That way the Repetier-Host will convert the units correctly into millimetres.

Object Placement

Objective

Prepare all objects you want to print, so your printer can print them. Learn how to arrange them on your print bed. Rotate and scale them to your preference.
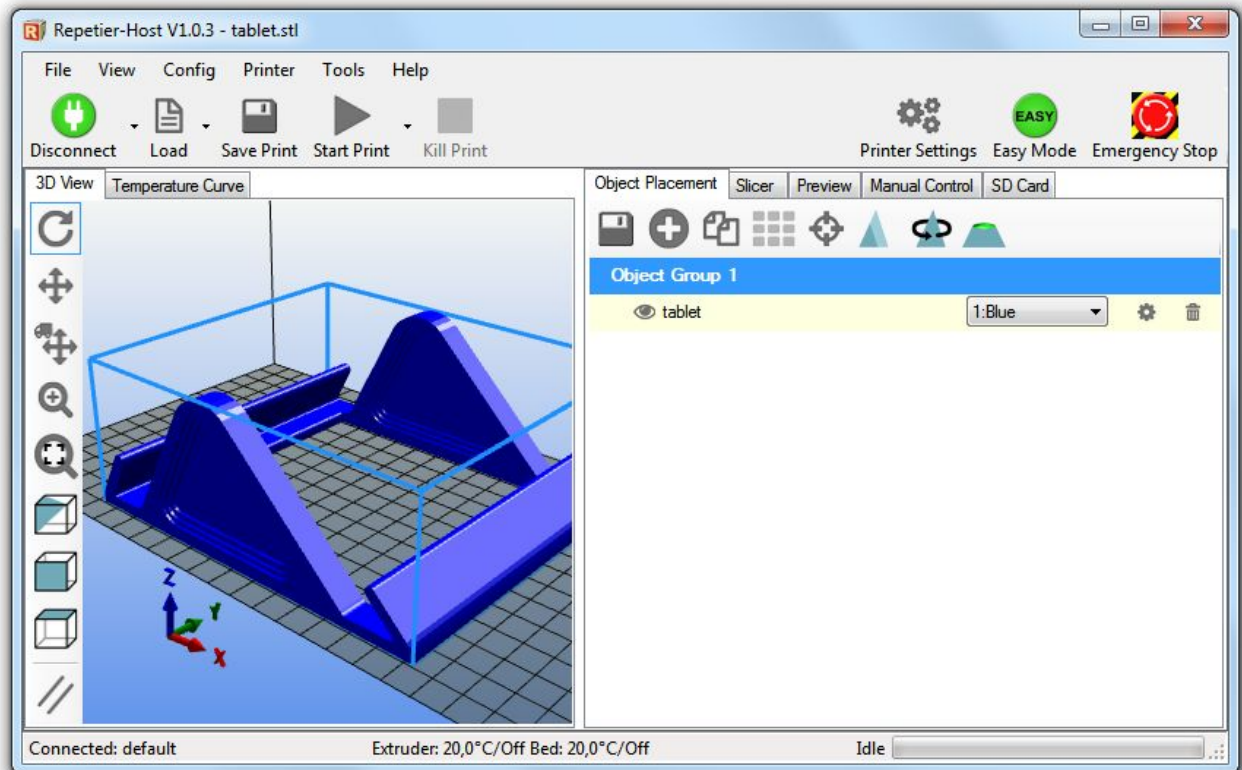
Workflow

Open the "Object Placements" tab. You will see the print area on the left side. You start with importing all STL files using the "Add Object" button ⊕ . You can select multiple object files at once if you want. This must be in one of the following formats: STL, OBJ, 3DS. The Repetier–Host will always try to position the objects in a non overlapping way onto your bed.

Browsing 3D View
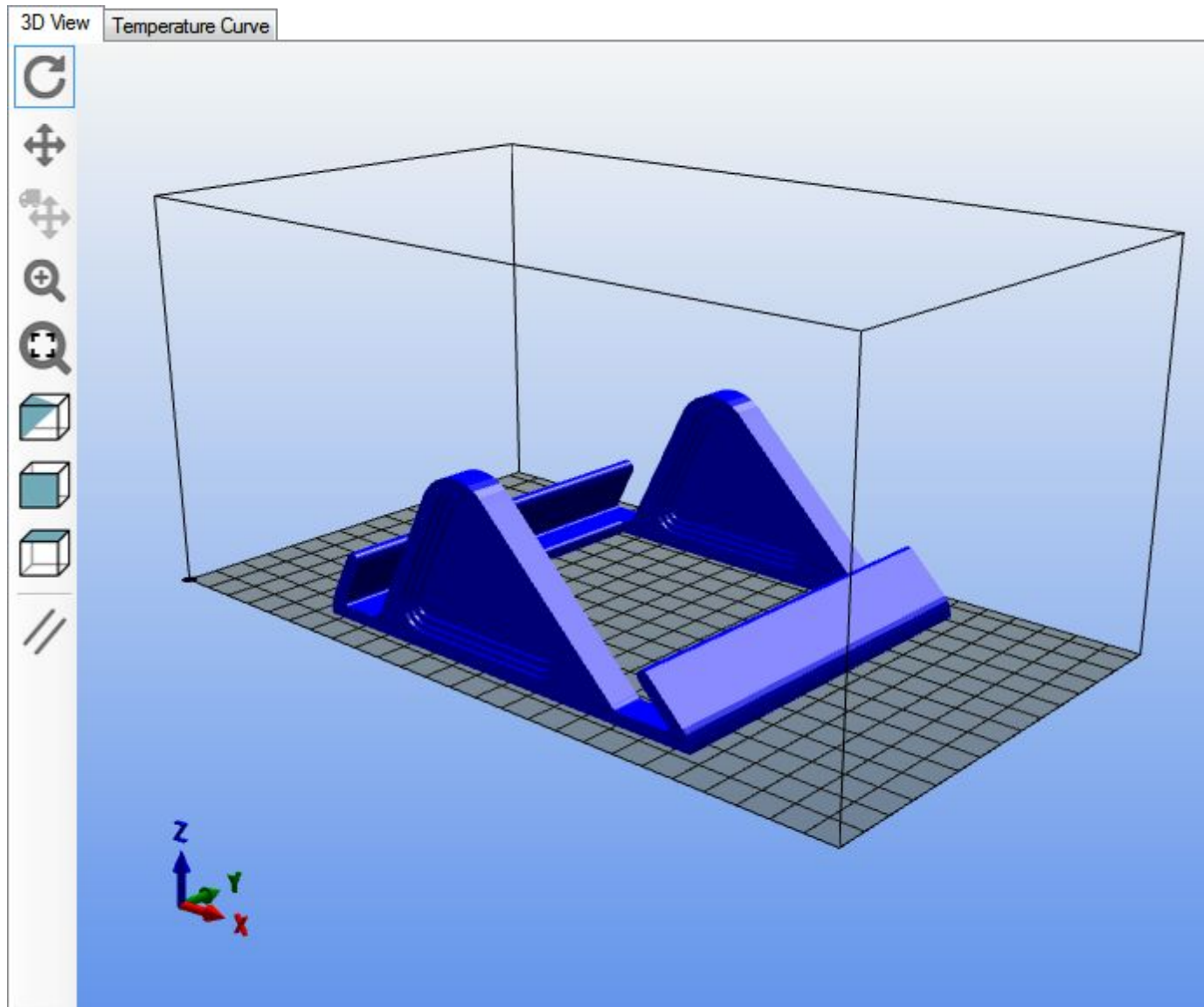
In this window you see your 3D objects. At the left side, you find some navigation buttons:

The first 4 buttons change the behaviour of the left mouse button. Starting at the top, you get

"Rotate" ⟳, "Move Viewpoint" ✛, "Move Object" ✛ and "Zoom" 🔍. All functions can be used with control keys, so that you don't have to change all the time.
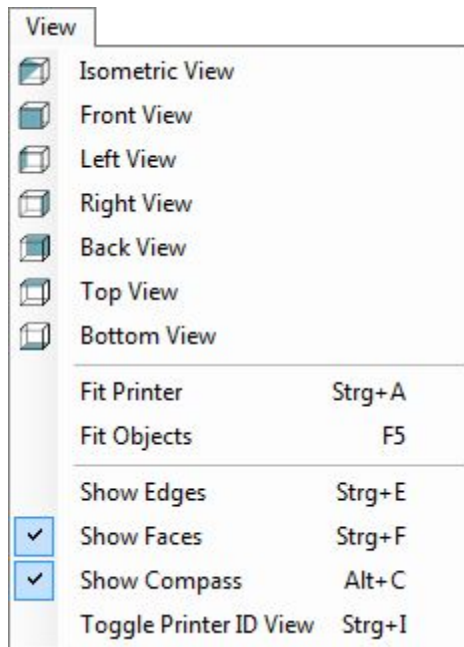
Control: Holding the control button, you can rotate the view with the left mouse button.
Shift: Holding shift button you can move the view point with the left mouse button.
Right Mouse Button: Move the object by holding the right mouse button.
Mouse Wheel: Zoom view

With this icon the objects will be zoomed to fit exactly in the preview area.

With the next three icons you can select between predefined view orientations. You find more view options at the top View menu:

Fit Printer (Strg+A): Zooms to fit the complete printer area into the window with maximum size.
Fit Objects (F5): Zooms to fit all objects into the window with maximum size.
Show Edges (Strg+E): Toggles display of triangulation edges.
Show Faces (Strg+F): Toggles display of triangulation faces.
Show Compass (Alt+C): Toggles display of coordinate compass at the left bottom.
Toggle Printer ID Vies (Strg+I): Toggles the display of the printer ID on top of the right tab. Use this to distinguish the printer when you have more than one Repetier-Host running.

"Use Parallel Projection" toggles between parallel and perspective projection.
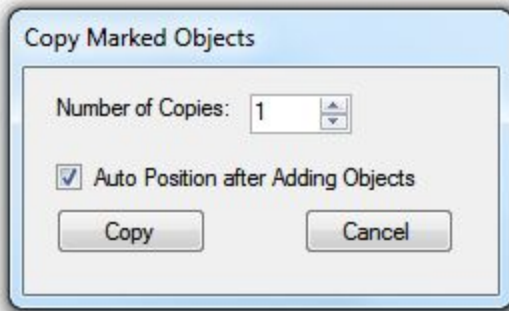
Object Placement

 Here you can export all displayed objects at once. If you save them as .amf file, the object grouping and material assignments remain intact, if you save it as .stl or .obj file, everything gets combined into one object.

 Here you can add objects in .stl, .obj, .amf and .3ds format.

 Here you can duplicate the marked object(s) as many times as you want.

 Click here to place all objects so that they fit on the bed.

 This function centres the marked object in the centre of the bed.

 With this function you can scale the marked object.

If the lock is closed, all axes are changed simultaneously. If you click on the lock to unlock, you can adjust each axis separately and the object may be distorted. A click on "Scale to Maximum" enlarges the object so that it has the maximum printable size.

 With this function you can rotate the marked object around each axis. Click on "Lay Flat" to place the object flat on the bed.

This option is only used to control the object and has no influence on the print. With "Position" the height of cut position is defined, "Inclination" and "Azimuth" define rotation of the cutting plane.

Cut Objects

Position

Inclination

Azimuth

Selecting and moving objects

You can select one object by right clicking on it. If you press the ctrl-key while right clicking the selected object is added. ctrl-clicking a selected object removes it from the selection.

To move the selected objects, hold the alt-key down while left clicking and dragging the objects.

When you select "Top View" from the left toolbar, the movements will follow your mouse movements. If you don't, the direction of the move is not the same as the mouse movement.

Depending on your configuration, the object may start to pulse or change its colour, if it is not entirely on the print bed. This should help you to identify printing problems, even before you start slicing.

Grouping Objects

Object groups are used only for multi extruder prints. For multi extruder prints you normally get one stl file for each colour. After you have loaded them, each has its own group, which normally causes wrong relative positioning. So you need to drag the second stl onto the first stl to merge them into one group. After merging you have to assign each file a separate extruder. Assigning the same extruder will normally cause problems during slicing.

Now it's time to slice your objects. Open the slicer tab and continue there. More on this in the next chapter.

Slicing

Setup your slicer

Kora recommend you use the Cura Slice Engine to slice your models prior to printing. Select **Slicer: 'CuraEngine'** from the drop-down menu below
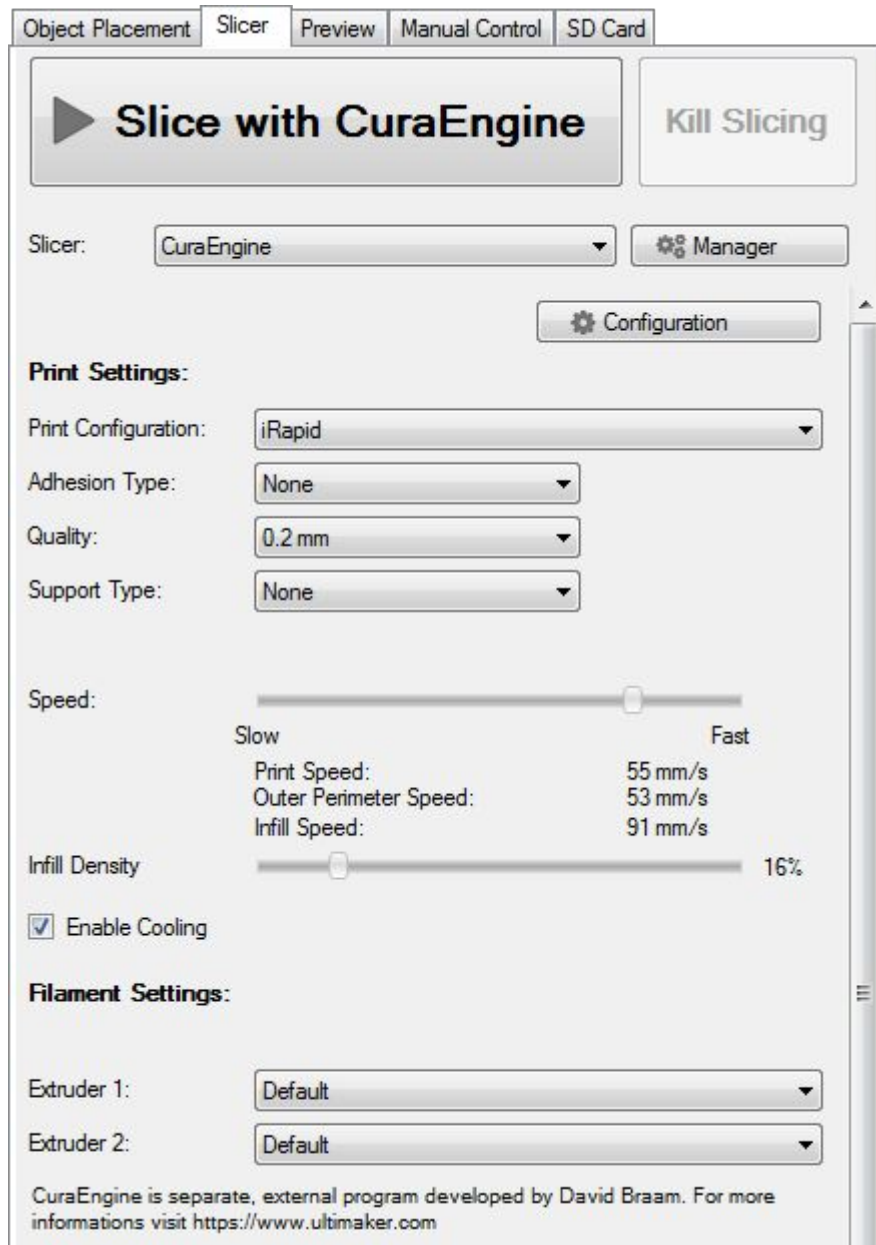
CuraEngine is external slicer software which is bundled with the Repetier-Host. Here you select your printing configurations to use plus some additional important settings. To define the parameters behind your configuration, click the "Configuration" button to enter the setup.

Kora supply 'MIDI' pre-configured settings for the Cura Engine on your Kora USB stick supplied with your machine or to download from the Kora website www.kora.co.uk/downloads

These pre-configured settings are a 'starting place' for your Kora MIDI Cura slicing settings. As you become more familiar with the 3D Printing environment, you will from time to time change these settings to 'fine tune' individual prints and when printing with new filament materials.

To load the pre-configured settings select the 'configuration' button on the main Repetier-Host screen (see below – check the circled items are also selected as shown before proceeding)
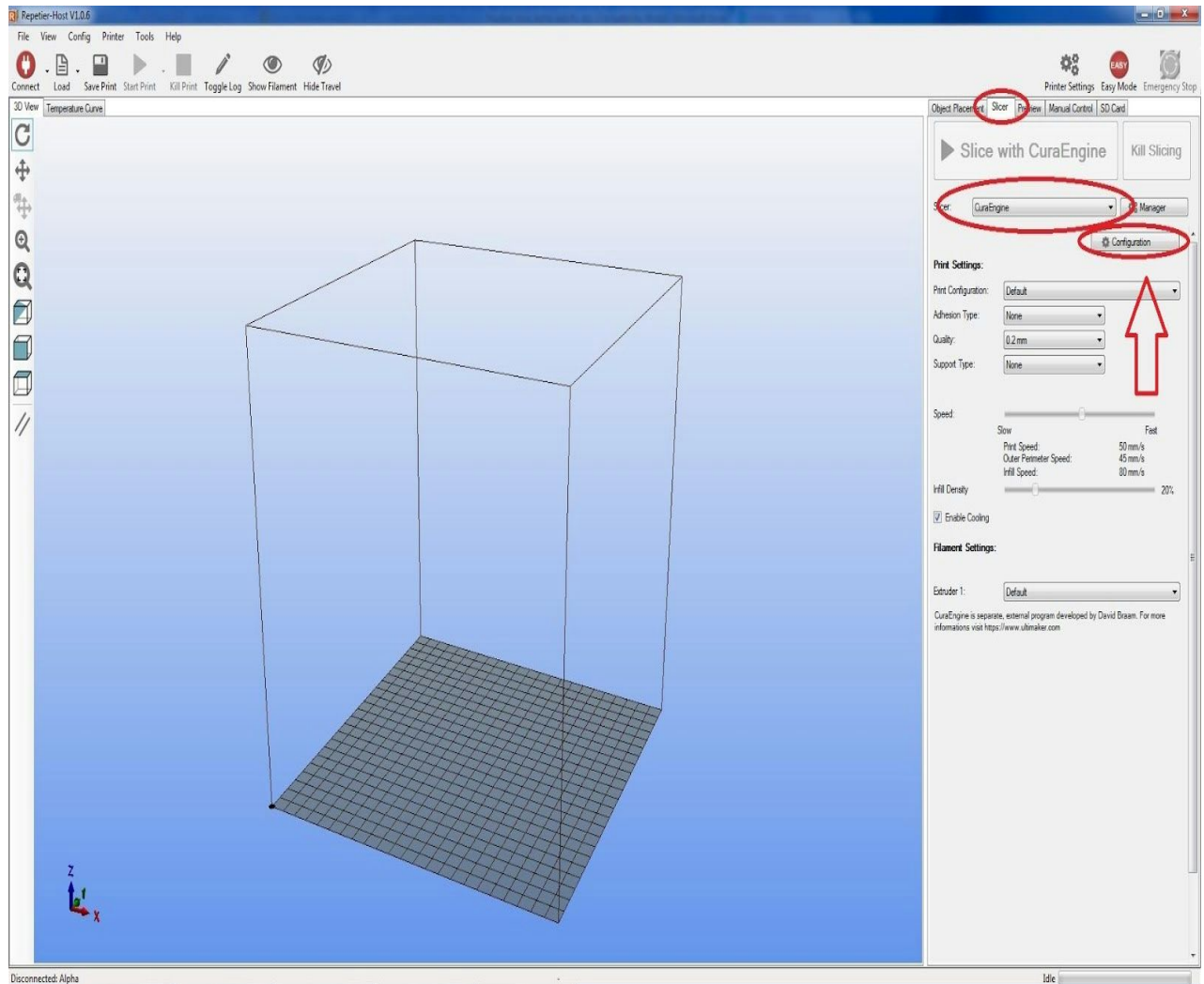
Then select the 'Printer' tab at the top left of the window (see below). Then click the 'import' button. The file menu will appear, navigate to your Kora USB stick or the folder you downloaded your Kora Cura Configuration files to and select the file 'Kora_MIDI_1.rcp'

Then click the 'save' button above and to the left of the 'import' button

Now select the 'Filament Tab' at the top left of the window (see below). Then click the 'import' button. The file menu will appear, navigate to your Kora USB stick or the folder you downloaded your Kora Cura Configuration files to and select the file 'Kora_MIDI_PLA.rcf' This file is for use when printing with PLA Filament. If you print with ABS Filament, you should also import the file 'Kora_MIDI_ABS.rcf'
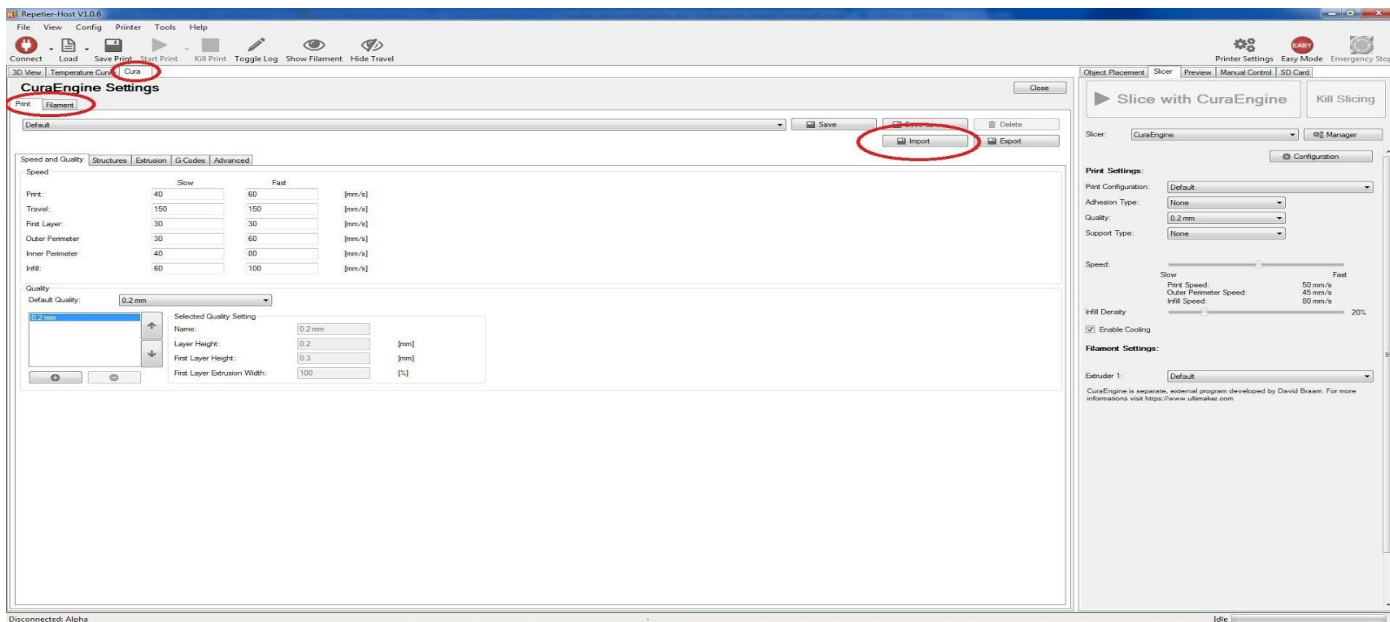

These files need only to be imported once. After that they will be automatically saved and available in the drop-down menu on the main Repetier-Host SLICER screen (see below) each time Repetier-Host is opened. Usually, select **Print Configuration:** as Kora_MIDI_1. Always select the appropriate **Filament Settings:** as Kora_MIDI_??? – ??? is for each different filament type you are printing (eg. Kora_MIDI_PLA for PLA filament or Kora_MIDI_ABS for ABS filament etc. – this selects, amongst other features, the correct extruder temperature for printing your chosen filament.
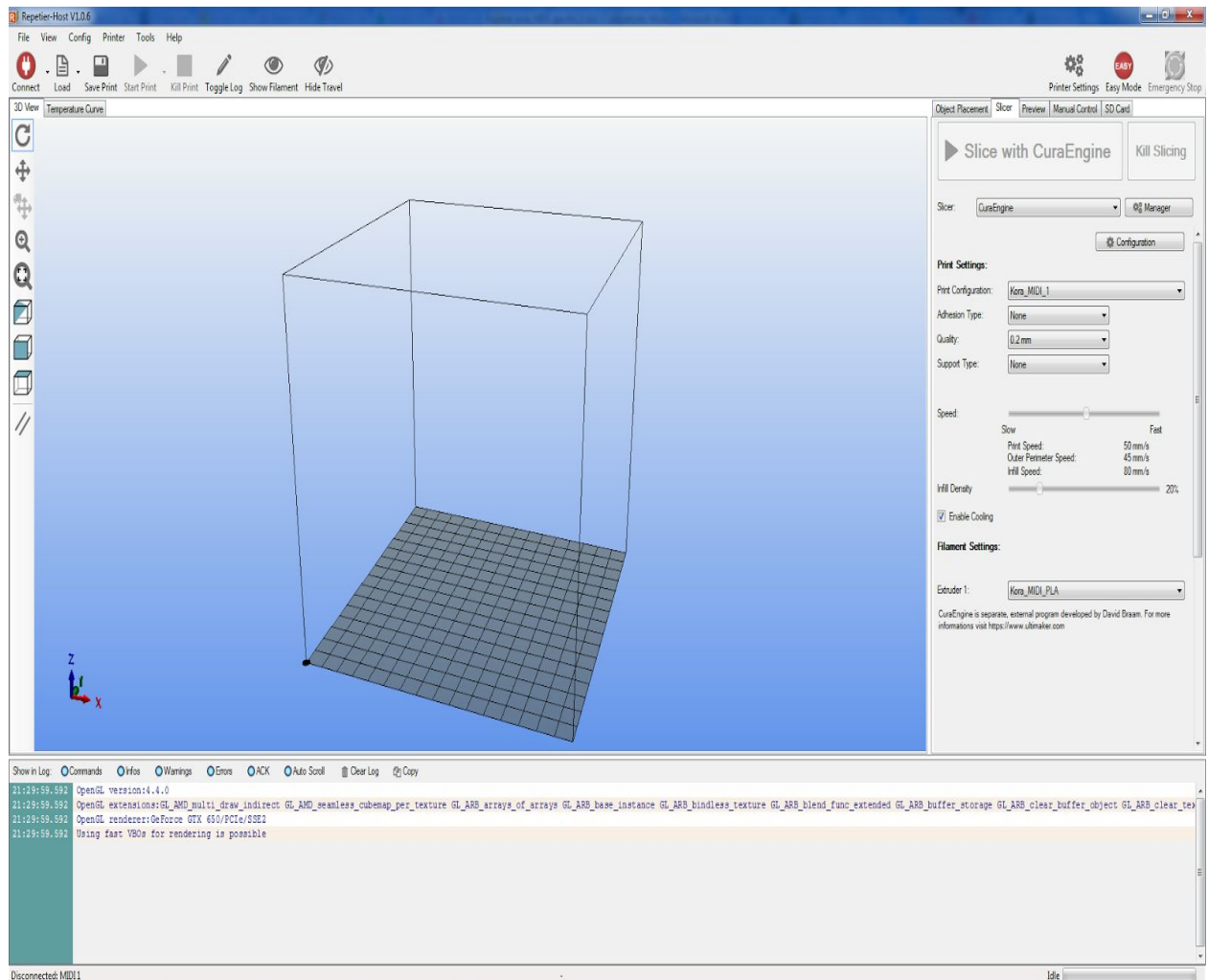
You can select any 'control option' in Repetier-Host, then hover your mouse over that setting and an 'explanation bubble' will appear guiding you how to use this option (eg. 'Adhesion Type', 'Support Type' etc etc.) This feature is particularly useful as it refers specifically to the control button or bar you have selected. Many controls throughout Repetier-Host have this feature! (see page 50 below for example)
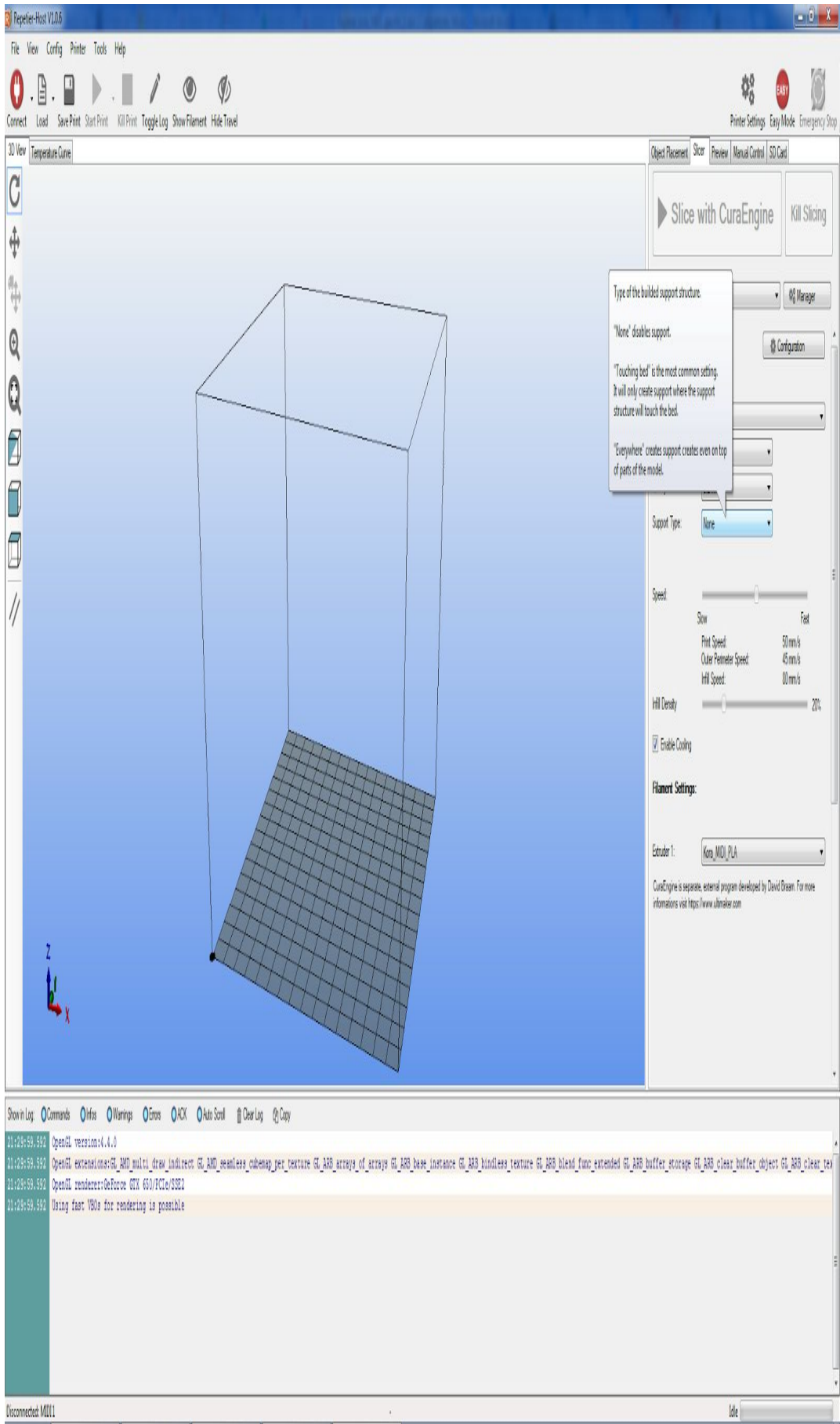
With reference to the 'Quality' drop-down box, this controls the print resolution. 0.2mm is deemed standard resolution, 0.4mm is course (sometimes referred to as 'draft' quality) and 0.08mm is fine quality. The finer the quality the smoother the print finishes, but the longer it takes to print. 0.2mm is usually a good compromise between quality and speed of print and will be selected in most cases.

Manual Control
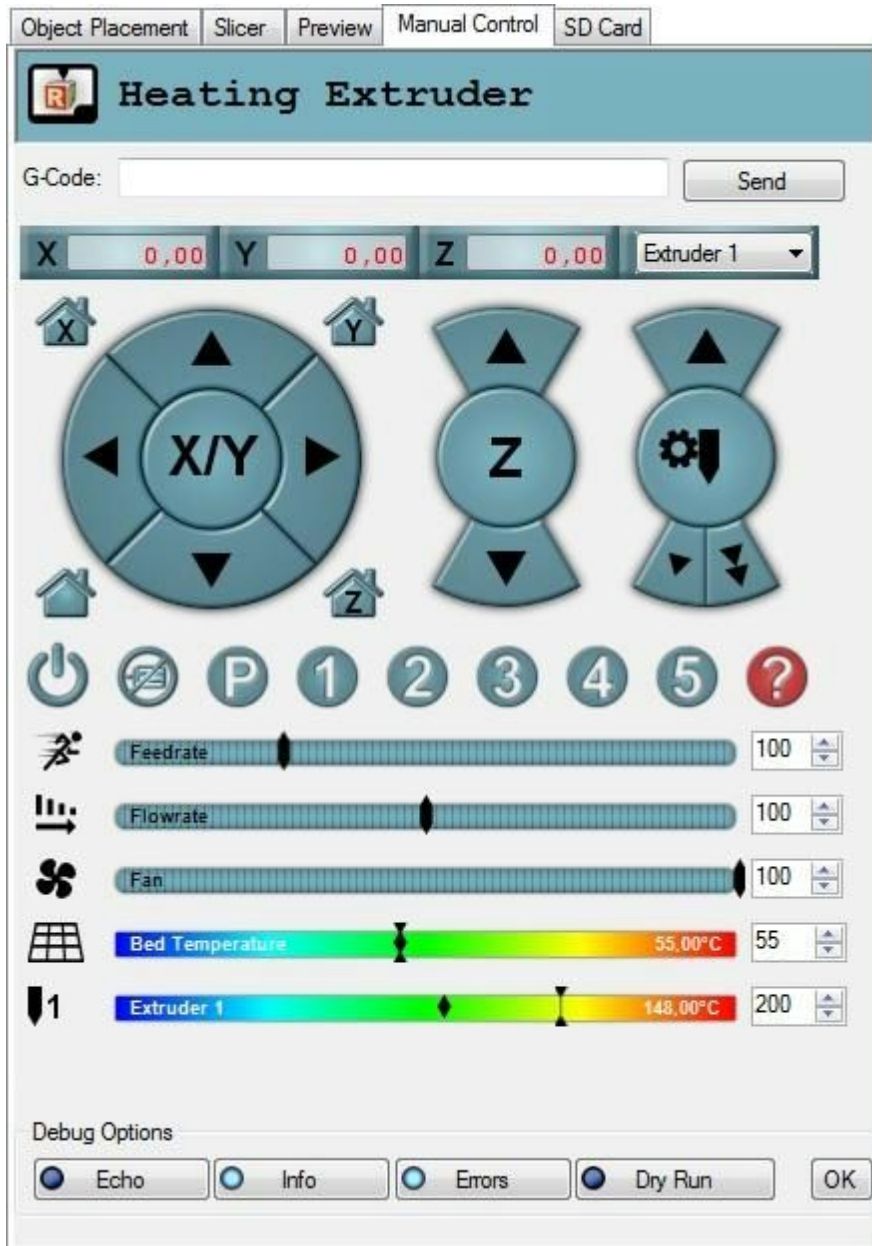
When you turn your printer on, you will often come to this tab. At the top you see the most important printer status, so you always know what is happening.

The next row allows it to send any g-code command you like. Enter it and press return or hit the send button. With the cursor up/down keys you can move through the history of manually sent commands. If the "Easy Mode" is enabled, this field is not displayed.

G-Code:		Send

The next block controls the positioning of the extruder. With the arrow keys you move the extruder in any direction. When you hover over the arrow, you will see a distance appear in mm, telling you how large the move will be. At the top you see the current extruder position. After the connection, they are red. Red means the Repetier-Host has no idea where the extruder really is. Press the home button to move the extruder to its defined position. After that the colour turns black, telling you the position is known. From now on, the moves are only possible inside the printer cube defined in the printer settings. So if you are at x=130 and the cube is 158 mm wide, pressing 50 mm right will only move to 158 mm.

X	0,00	Y	0,00	Z	0,00	Extruder 1
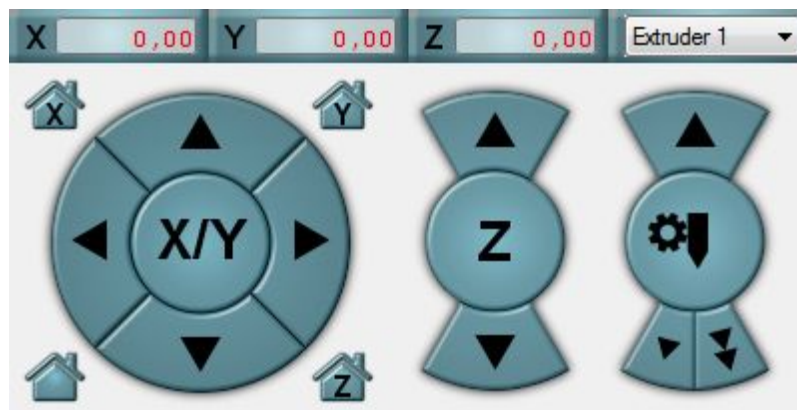
CAUTION: This protection works only for the arrow keys. Below the arrows you have the following buttons:

Power: This turns the power supply on. This is not supported on Kora MIDI

Stop motor: Will disable the stepper motors.

Park: Moves the extruder head into parking position defined in printer settings.

Help bubbles: When you activate this, you will get help bubbles for all fields of the manual control.

Buttons 1 – 5 are for advanced users.

The "Speed multiply" slider allows it to change the printing/move speed in relation to the send feed rate. This function is compatible with the Kora MIDI. Be careful if you increase the speed. Very high speed can deteriorate print quality!

Feedrate	100

If the "Easy Mode" is disabled, you can change the flow rate, i.e. the amount of the extruded filament. A higher value gives a wider printed line.

Flowrate	100

The extruder and print bed blocks allow it to manually change the temperature during a print. The temperature can be set either on the right in the text field as well as by clicking on the temperature curve. If you change it in the text field, you need to press return or leave the field to set the value. By clicking the extruder or bed icon the extruder or the bed is deactivated or reactivated. In the right area of the temperature curve you see the last read temperature (you have enabled automatic temperature reading in printer settings?).
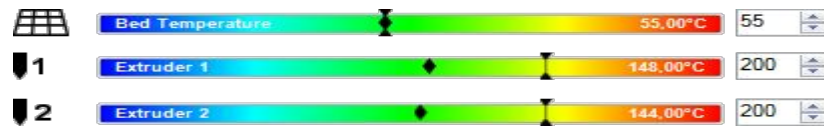
Here, you can change the PLA cooling fan speed and activate/deactivate it the same way as the temperatures.

In the last row you can set debug options for the printer if "Easy Mode" is disabled. Echo repeats the received lines, so you normally want it turned off. Info and error show messages at that debug level. These are interesting, so leave them on. The last one is something special, which works up to now only with Repetier-Firmware. In dry run mode, the firmware will ignore all commands to set temperature or extrude. That way you can send a file without using any filament. The last button "OK" re-sends the start/send command to the printer and is not normally used.

Linux Only Installation (non Windows)

(SKIP TO PAGE 3 – if installing on a Windows Computer)

The linux version comes as gzipped tar file. Move it to where you want your files and unpack its contents and run the post installation script:

tar -xzf repetierHostLinux_1_03.tgz

cd RepetierHost

sh configureFirst.sh

After that you have a link in /usr/bin to the installation, so you can start it with Repetier Host. Make sure you have all required Mono libraries installed. If you are in doubt, install Mono develop, which has all needed libraries as dependency.

One problem that most linux distributions have is, that the normal users are not allowed to connect to a serial console. You need to put your user into the right group. On Debian you can call:

usermod -a -G dialout yourUserName

to add your user to the group dialout.

First start

After the first start-up, a dialog may pop up asking you, where your work directory should be. The latest windows versions skips this step, because the installer already created a work directory for you. You can select any directory, where you have write privileges, but it is advised to use a separate directory for this purpose.

The work directory is, where the Repetier-Host will put temporary stl files, sliced results and if enabled the log file.