Tips & Tricks

General

Here you will find information about general issues concerning the operation of the RepRap Industrial in your specific work environment.

Commandline Access to the linux operating system via SSH

Use SSH on your computer connected to the same LAN as your 3D printer to log in to the RepRap Industrials' built-in BeagleBone Black. You can use the hostname from the printers' Backend-URL and log in with the following access data:

User: kiosk Password: eight-digit combination from the serial number at the back of the device. Take the first two four-digit blocks - XX-AAAA-BBBB-CCCC-YYYY becomes an "AAAABBBB" password.

Setting a static IP address for RepRap Industrial ethernet connection

First, establish a commandline connection to the printer.

From within the terminal session, edit the network configuration via the command line editor "nano"

```
sudo nano /etc/network/interfaces
(use the same password as for the SSH connection)
```

The current DHCP setup looks like:

```
# The primary network interface
auto eth0
iface eth0 inet dhcp
pre-up iptables-restore </etc/iptables.rules</pre>
```

Change the setup according to your needs. Example:

```
# The primary network interface
auto eth0
iface eth0 inet static
address 192.168.1.20
netmask 255.255.255.0
network 192.468.1.0
broadcast 192.168.1.255
gateway 192.168.1.254
dns-search family.local
```

dns-nameservers 192.168.1.254
 pre-up iptables-restore </etc/iptables.rules</pre>

The rest of the file remains unchanged. Save the file using CTRL+X and confirm the overwrite query with "Y". Disconnect and finish by typing

exit

Shut down (see Manual control) and reboot (power-on button, 11 in fig. 4) the RepRap Industrial to establish the alterations.

Use a custom NTP server for time signals

First, establish a commandline connection to the printer.

From within the terminal session, stop the NTP daemon background process

sudo service ntp stop
(use the same password as for the SSH connection)

Edit the NTP daemon configuration via the command line editor "nano"

sudo nano /etc/ntp.conf

Search for the few lines beginning with

server ...

and add an additional new line before these with the address to your local NTP server like this

server 192.168.1.123

add another additional statement anywhere in this file

```
# ignore panic threshold for huge time differences tinker panic \boldsymbol{0}
```

The rest of the file remains unchanged. Save the file using CTRL+X and confirm the overwrite query with "Y".

Re-enable NTP client service for background operation

sudo service ntp start

Disconnect and finish by typing

exit

Shut down (see Manual control) and reboot (power-on button, 11 in fig. 4) the RepRap Industrial to

establish the alterations.

Changing Shell Password in the BeagleBone Black Operating System

After manually creating a new Micro-SD Card from a pre-packaged upgrade release provided by *Kühling&Kühling*, the user account running the RepRapOnRails software in the Linux operating system will be in default configuration. To change the password, use SSH on your computer connected to the same LAN as your 3D printer to log in to the BeagleBone Black. You can use the hostname in the printers' Backend-URL as its address and log in with the following access data:

User: kiosk Password: kiosk

Now you can set a new password by entering

passwd

and following the instructions. In delivery condition the unique password is an eight-digit combination from the serial number at the back of the device. Take the first two four-digit blocks (example: XX-AAAA-BBBB-CCCC-YYYY becomes an "AAAABBBB" password).

Slicing

Any information we gather related to improving or easing the slicing process is listed in the following paragraphs.

Profile display problems in Slic3r for Windows

This bug will be fixed with the upcoming release of SLic3r v1.2.x.

It is a known problem that in Slic3r for Windows the drop-down lists for the profile selection are too narrow to display the full profile name. This can make it impossible to choose the correct profile when processing your slicing settings for creating a G-code.

		Sick	- 0
in Plate Neider Poly	Lations Analyziational		
manufactul shids a	Distant conducts		
default - latting/illusting Testing	Beilige	at 200 pt 100 mm	
attingdiadday hplig	Print carries	a MJ 5/200 mm	
attingfilleddau Daglag mys farse	Zome	8	
pageotation shots good	Acres 1		
	R-sole favor	Replac dilatin/fprime:Repeter: +	
	the relative & detromat		
	Capitilito		
	Educion	1 0	
	Advanced		
	Dia fermani stradam	1 10	
	Tribution levels	2 10	

Narrow menus make it impossible to read the profile names in Slic3r v1.2.0 for Windows

Try the following to bypass the problem:

- Select a profile from the drop-down list and click the <save> symbol. This way, you make the profile name visible and editable.
- Delete the current profile name and replace it with a short description, e.g. "LEFT ONLY" instead of "Kuehling&Kuehling RepRap Industrial -LEFT EXTRUDER ONLY".
- The shorter description will be fully visible in the list.

Repeat these steps for all long profile names.

Kuehling&Kuehlin 🗧 🙀	Size and coordinates	
di Ceneral	Bed size:	x 200 y 180 mm
© Custom G-code	Print center:	x: 100 y: 90 mm
T Extruder 1	Zofftet:	a mm
	Firmware	
	G-code Havor: . Save	
	CONTRACTOR Save priv	Ner settings as:
	Capa	
	Extruders:	Abbrechen DK
	Advanced	
	Use firmware retractions	
	Vibration limit:	2 16
Plate Priot Settings Pf Exec Priot Settings Pf Executing Execution 2 III Controls Conde III Controls Conde III Extrader1	Bed size: Print center: Zoffset: Firmware Goode filavori © See	x 200 y 100 mm (300 y 10) mm 0 mm
Plater Print Settings Fil	ament Settings Prinker Settings	
	Size and coordinates	
LEFT ONDE : 🗎	Bed size:	x 2002 y 1801 mm
		without all react and
Cerent	Print center:	a: 1001 9: 9990 mm
@ Control @ Custom G-code	Print center: 2 offset:	a: 1001 yc 1000 mm
@ Control @ Custom G-code	Print center: Z offset: Firmware	x 1307 y 6000 mm 800 mm
@ Control @ Custom G-code	Print center: 2 offset: Firmware Goode Revor:	a: 1001 yc 1000 mm
@ Control @ Custom G-code	Print center: Z offset: Firmware	x 1307 y 6000 mm 800 mm
@ Control @ Custom G-code	Print center: 2 offset: Firmware Goode Revor:	x 1307 y 6000 mm 800 mm
@ Control @ Custom G-code	Print center: 2 offset: Firmware G code flavor: Use relative E distances:	x 1307 y 6000 mm 800 mm
@ Control @ Custom G-code	Print center: Zoffhet: Firmware Goode Renor: Use relative E distances: Capabilities Extrudent:	s (1007) 9 (1000 km 000 mm Regular (Monto/Sprinter/Nepstor)
@ Control @ Custom G-code	Print conter: Zoffnet: Framwere Goode Rimor: Use relative E distances: Capabilities Katruders: Advanced	a tool y leen an oo m Replus (Merin/Sprinter/Reprint) = 1
@ Control @ Custom G-code	Print center: Zoffhet: Firmware Goode Renor: Use relative E distances: Capabilities Extrudent:	s (1007) 9 (1000 km 000 mm Regular (Monto/Sprinter/Nepstor)

Overhang - adjusting the layer thickness

If you want to print filigree objects with overhanging structures without adding supports, try reducing the layer thickness for this print by 15 - 20%. This will result in a finer Z-axis resolution and increased overlay of subsequent layers so that more stability is gained over the height of the target object.

됩

Modifying infill and perimeters

There are currently two Slic3r profiles available at our GitHub repository that provide preset ready-toprint slicing settings. Her are some tips for handling these profiles to adjust them to your needs. The *SOLID* profile normally needs no modification since it comes with stable, reliable settings for printing solid objects with 100 % infill.

The *ECO* profile supplies settings for objects with loosened infill. This makes objects lighter and reduces the print time and the material consumption. To achieve good results, the settings may have to be adjusted.

We recommend to set the following always for printing ECO objects:

- A hull thickness of 1.0 to 1.5 mm. As a rule of thumb divide the hull thickness by the nozzle tip diameter and set the amount of perimeters equal to the result. Such, a closed, smooth and stable hull is ensured.
- Honeycomb infill as it provides highest stability at optimal density.
- A minimum infill of 15 %.
- A maximum infill of 30 %. Increasing the infill further does not have a significant advantage compared to a solid body but strongly increases the printing time.

Operation

In the following you will find more detailed descriptions of functions of the RepRap Industrial. In most cases, these are topics upraised by support requests or in-house lab experiments.

Calibrating the extrusion

The stability and dimensional accuracy of any printed object require a correct amount of filament conveyed through the nozzle. Too little extrusion and the part will be thin-walled, fragile and likely to break. Too much material is likely to clog the nozzle and ruin the print. The amount of material effectively conveyed through the nozzle is depending on:

- the 3D printer itself slight variations are possible due to the manufacturing process;
- the actual filament diameter in the range of the dimensional stability of the filament;
- the printed material's properties the extruder drive wheel grinds deeper into softer materials, thus reducing the actual diameter;
- the idler tension high tension will cause the drive wheels teeth to grind deeply into the filament, causing a dilation of the material and a reduced output.

To make sure that the print result is stable and accurate, an *extrusion multiplier* must be found for every material on every apparatus; it may be that this factor must be found for every spool of filament.

The correct extrusion multiplier is set in the slicing software, compensating for the above named variables.

To find the correct multiplier, open the *Setup* menu at the GUI and choose [Calibrate Extrusion].

A detailed description for ABS filament can be found here.

For other materials, the same procedure applies but you have to prepare a G-code for the test print.

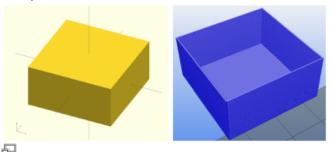
To create a G-code for the extrusion calibration,

dimensions 30x30x15 mm (download here).

you need an stl-file of a cube with the

Idle		Emergency STOP
Print Configuration Setup Exper	Control Log	
Wizards Print Bed Leveling	Nozzle Change	Vieb Interface UI http://rficbadofi445
Unload Filament Load Filament	Calibrate Extrusion Calibrate Extruder Offset	Software Versi RepRapOnHails v1.3 Hardware Revisi

Choose the [Calibrate Extrusion] wizard from the *Setup* menu (either software version).



The 30x30x15 mm calibration cube as

Load the stl-file in Slic3r and select the Print Settings tab.

OpenSCAD model and ready sliced with Slic3r (view taken with Repetier-Host).

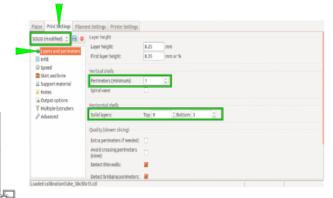
Add	Gelete Delete All	Accange	More	Fever	45° ccw	9 45° cw	9 Rotate	Scale	- MR Split	View/Cut	O Settings.
			Name		Copi	e Scale				Print set	tings:
			calibration	cube_30x30x	15.38 1	100%				ECO	-
										Filament	
										Kuehān	gakuehli ;
9	_									Printers	
- 553 										singleE	druderN :
										Export	G-code
										Export	STL
			info								
	X = 140			10.00 x 30.00 x	: 15.00			Volume: 1	3500.00		
			Facets: 1 Manifold: 1	12 (1 shells) Yes				Materials: 1			
orded call	brationCube_30x30x15.	ut l									

Load the calibration cube stl-file in Slic3r.

Choose the SOLID profile as a basis.

Choose the following settings to make the cube a box without a lid and only one perimeter of 0.5 mm thickness for a wall:

Layers and Perimeters → Vertical shells Perimeters (minimum)	1
Layers and Perimeters \rightarrow Horizontal shells Solid Layers TOP	0
Layers and Perimeters → Horizontal shells Solid Layers BOTTOM	3
Infill → Fill density	0 %
Advanced \rightarrow Default extrusion width	0.5



Select the Layers and perimeters according to the adjacent table.

OLID (modified) 🚦 🔒 🚇	arit		
Layers and perimeters	Fill density:	0 + %	
	Fill pattern:	rectilinear	
© Speed	Top/bottom fill pattern:	rectilinear	
Skirt and brim	Reducing printing time		
Nobes	Combine infil every:	1 layers	
Output options	Only in fill where needed:		
Advanced	Advanced		
	Solid infil every:	0 lapers	
	Fillangle:	0 1	
	Solid infil threshold area:	70 mm ²	
	Only retract when crossing perimeters:	8	
	Infil before perimeters:		

됩

Select the Infill according to the adjacent table.

· C Slate				
Plater Print Settings (10)	enest Settings Printer Set	cings.		
calibrationCubeSe : + +				
B Loyers and perimeter	the fault extrusion width:	0.5	mon-or to (leave Orlar auto)	
E offi	110100	15070	THE REPORT OF COMPANY	
🖀 tikir Land brim	Perimeters	0	mm or % (leave Orlan default)	
🚊 Support motorial	External perimeters:	0	mm-or % (leave 0/or default)	
O speed	148	0	ment or % (leave Offer default)	
T Multiple Draviders	tolid infit:	0	mm-or to (leave offer default)	
Quipat aptions	Top solid infil:		mm-or % (leave Offer default)	
I Plotas	support material	0	mm or w (leave other default)	
	flow			
	thridge from ratio:	1		
	Other			
	XV Size Compensation:			
	Threads:	2	1	

Select the Default extrusion width according to the adjacent table.

INFO



Saving these settings as an "extruder calibration" profile will make this calibration much more



comfortable in the future.

Upload the G-code to your 3D printer, print it, measure the wall thicknesses, and calculate the mean value.

Then open the [Calibrate Extrusion] wizard and enter the mean value.



Adjust the preset "0.50 mm" value via the touch buttons and read the necessary extrusion multiplier directly from the display.

The displayed extrusion multiplier can be entered in Slic3r (Filament settings) and saved in the filament profile.

Enter the extrusion multiplier in Slic3r and save the filament profile (rename!).

Deactivating the heating elements after end of print job

Sometimes you may want to start a print job just before finishing time or the weekend. Since there is currently no automatic shutdown function, the 3D printer will then stay on all night respectively some days. With the following description you can alter the End G-code of your print so that the heating elements are shut off after the print job has been finished so that the power consumption is reduced significantly. A side effect is, that due to the fact that the build chamber needs some hours to channel off the heat the cooling process is slowed and thereby internal tensions of the printed object are reduced. To deactivate the heating elements after a print job:

- Open the [Printer Settings] tab in Slic3r and choose the *Custom G-code* menu.
- Activate the End G-code editor by clicking into the text field.
- Position the cursor in the last line before the ; /END-GCODE entry.
- Enter the command M104 S0 T2 This will set the heating elements of the build chamber to a temperature of 0 °C.

KuehlingbKuehlin; : 🔝 🛛	Start Goode	
Ceneral Castan Geode T Extruder 1	SUNAT-GCODE ALC : activate build chamber GA 12 : seal 2 seconds GR : use absolute coordnates GZB : heare all axes	
	Earl Coulde DAYS 00 LOSS 1010 59 TO 1010 51 TO 100 51	
	Tool change G-code	

Switching off the build chamber heating

 Enter the command M140 S0 This will deactivate the print bed as the last action of the current G-code. elements by modifying the Slic3r custom End G-code.

Adjusting the build chamber temperature

NOTICE

The build chamber temperature is preset to the maximal permissible temperature of +70 °C at delivery.

Exceeding +70 °C will damage interior components of the RepRap Industrial such as stepper motors, bearings and electronics.

Since hitherto the RepRap Industrial is the only commercially available Open Source 3D printer with an actively heated build chamber, common slicing software does not feature ambient temperature settings. For some materials, it is advantageous to modify the chamber temperature together with the other temperature settings. The build chamber's temperature of the RepRap Industrial is set via the "Start G-code" which can be manually altered.

	Filament Settings Printer Settings
LEFT EXTRUDER O 🔅	Start C-code G92 E0 ; reset extrusion distance To ; ensure that left extruder is selected
Custom G-code TExtruder 1	; PREHEAT BED AND CHAMBER M104 570 T2 ; set recirculating air heater to 70 degree celcius target hemperature
	C12100 ; move bed to chamber center for faster heating distant from air flow M190 Sflirst. laver. bed. temoeraturel : set hed temoerature and wait.
	End G-code
	; END-GCODE
	G92 E0 ; reset extrusion distance MB3 ; use relative distances for extrusion G1 F1440.000 E-1.5 ; retract lastly active extruder MB2 ; use absolute distances for extrusion
	C30 ; use absolute coordinates C32 E0 ; rest extrusion distance C28 · home all axes
	Before layer change G-code

To change the build chamber temperature (in the following example we use our standard *Slic3r* - other software may differ in denotations):

- 1. Open your slicing software.
- 2. Open the tab "Printer Settings" and choose "Custom G-code".
- 3. Go to the line reading:

M104 S70 T2; set recirculating air heater to 70 degree celcius target temperature

 Change the entry "Sxy" (here S70) by replacing the value xy with the desired temperature, for example 50° C:

M104 S50 T2; set recirculating air heater to 50 degree celcius target temperature Version 1.2.9 - Remember to check for updates at http://slic3r.org/

됩

Changing the build chamber temperature by modifying the Slic3r custom Start G-code.

(for logical reasons, the comment should be aligned)

5. If you want to keep the settings, save them in the profile (see Slic3r manual).

Any G-code exported with this profile loaded will heat the build chamber to the stated temperature prior to printing.

G-code manipulation at the GUI

The following list contains supported G-code commands that can be used on demand to directly interfere with a print procedure or setting via the G-code keyboard of the GUI's **Log** menu.



Ð

The G-code keyboard in the Log menu provides all keys to enter G-code commands.

Command	Effect	Example
G1	Coordinated Movement X Y Z E	G1 X130 Y85 Z1.75 E4.35
G4 S <seconds></seconds>	Wait for given duration in seconds	G4 S5 (waits 5 seconds)
G28	Home all axes	
G90	Use absolute coordinates	
G91	Use relative coordinates	
M80	Activate build chamber	
M82	Set E codes absolute (default)	
M83	Set E codes relative while in Absolute Coordinates (G90) mode	
M104 S <temp> T<extruder></extruder></temp>	Set temperature without wait	Adjusting the build chamber temperature, Deactivating the heating elements after end of print job
M109 S <temp> T<extruder></extruder></temp>	Set temperature with wait	
M140 S <temp></temp>	Set bed target temp without wait	
M190 S <temp></temp>	Set bed target temp with wait	
M221 S <extrusion flow<br="">multiplier in percent></extrusion>	Increase/decrease given flow rate	M221 S95 → decrease flow to 95 % of g-code value
M220 S <print speed<br="">multiplier in percent></print>	Increase/decrease print speed of all drive speeds	M220 S95 → decrease print speed to 95 % of g-code value