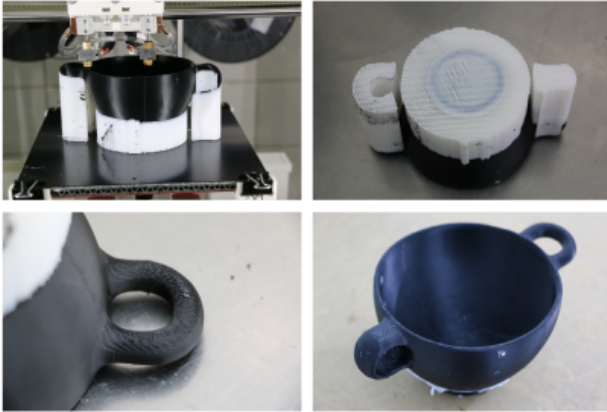


Dual extrusion



Excavation finding of the Archeological Institute of the University of Freiburg, reprinted on a HT500 with black ABS and HIPS as soluble support.

One of the HT500 3D-printer's core features is its dual extrusion functionality. Producing visually appealing bicolored or bimaterial parts, making functional parts out of a two-material combination or highly accurate complex geometries with fully attached soluble support structures become possible with the two hot-ends extruder head. And not only is the 3D printer equipped with the necessary hardware but also provides well-engineered software tools to ensure precise alignment of the hot-ends via the common easy-to-use wizards. Only load the desired combination of materials and perform the required calibrations, then prepare and upload your G-code and you are ready for dual extrusion. All necessary settings that are required for successfully printing with both hot ends are described in the following.

- [Basic settings](#)
- [Soluble support](#)
- [Post-treatment of soluble HIPS support](#)
- [Bicolored printing](#)

Basic dual extruder settings

The steps and settings described in the following paragraphs are valid for all multi-material print jobs. Specific information on [printing with soluble support material](#) or [bicolored](#) are described in the respective chapters below.



Things no. 67503 and 318057 from thingiverse.com, both printed as multicolor prints in red and white ABS on a HT500.

INFO



*The descriptions given in the following are only valid for hardware revisions later than rev1.3.0. Users of hardware revisions 1.3.0 and older need to equip their 3D printer with the **wiper upgrade kit** to participate in the latest developments. The upgrade kit can be ordered pre-assembled with all necessary parts via sales@kuelingkuehling.de. The dual printing profiles released for Slic3r v1.2.8 (and later) are exclusively configured for the use with the wiper.*

3D printer preparation

Depending on the task, the setup of the 3D printer for dual extruder prints may vary.

Printing bicolored objects will mostly be done with the same resolution requirements throughout the print, so installing the same nozzle diameter on both hot ends is recommended.

For printing support structures with a second material, printing the support with a larger nozzle helps reducing printing time since the printing resolution of the support is not vital for the appearance.

Slic3r will calculate interface layers accordingly for nevertheless satisfying results.

Make sure to have installed the nozzle suited to the task on each hot end, and that they are precisely leveled. Compare your setup with your Slic3r settings and adjust these if necessary.

Perform each of the wizards once before the first print. The settings may remain unchanged if no hardware changes (i.e. nozzle or barrel exchange) are made.

1. For dual extruder prints it is vitally important that both hot ends are precisely leveled to the same height. If not yet done, run the [Print Bed Leveling] wizard and follow the on-screen instructions.
2. The offset of the hot ends must also be calibrated to ensure accurate relative positioning. Run the [Calibrate Extruder Offset] wizard in the *Setup* menu of the GUI. Read and enter the values for X and Y offset and [Save] them.
3. Do not forget to check the [extrusion multiplier](#) for the installed materials and respectively to run the [Calibrate Extrusion] wizard for each hot end if required.

The 3D printer hardware is now ready for dual extrusion.

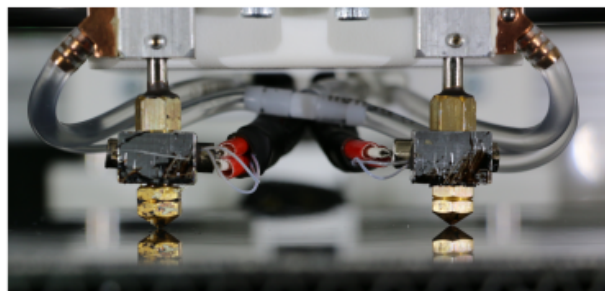
Wiper check

As previously mentioned, the HT500 dual extruder function is exclusively designed for the use with a wiper. The *Wiper Upgrade Kit* is available for 3D printers purchased before July 2015. Hardware revisions later than Rev1.3.0 can be equipped on request with the wiper ex factory.

The following assumes that the wiper has already been installed according to the assembly



Leveling and offset calibration are done via the GUI. Both only needs to be done once if no hardware changes are performed.



Precise leveling of both hot ends is mandatory for dual extruder prints.

instructions provided at delivery. If otherwise, find the [instructions](#) here.



The Wiper Upgrade Kit provides the necessary hardware for successfully printing dual extruder print jobs.

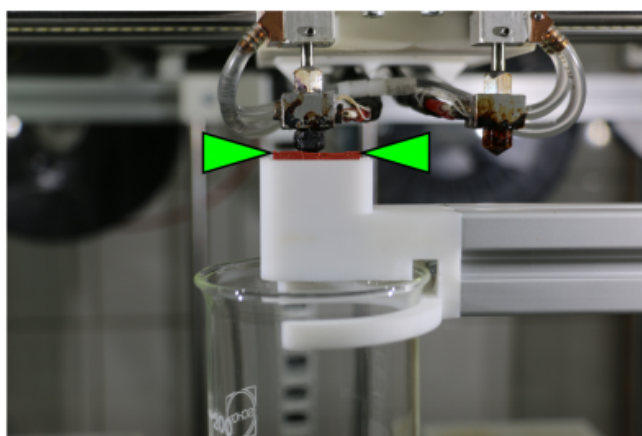
Check that the wiper lip and the nozzle tips are aligned correctly:

- Open the [\[Expert Control\]](#) screen and move the print head to the maintenance position.
- Move the left nozzle 30 mm to the right and 20 mm to the front via the movement control buttons. It should rest directly behind the wiper lip.
- Make sure the lip's upper rim is aligned in the middle of the nozzle's conical tip (see adjacent picture). Adjust the wiper lip if required.
- Home the print head.



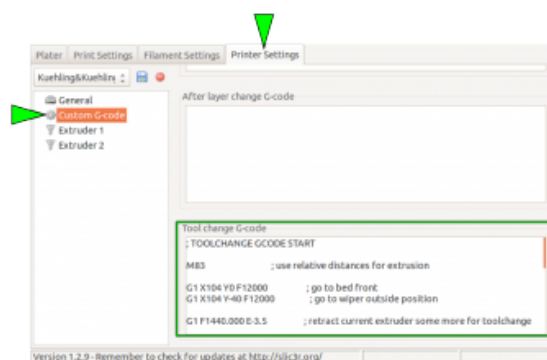
INFO

The nozzles must be precisely leveled to the same height.



Nozzle cone and wiper lip are correctly aligned.

All necessary settings to effect the required movement of the print head into a g-code are implemented in the Tool change G-code in the Custom G-code menu of Slic3r's *Printer Settings*. It is only filled in when one of the [Dual Extruder](#) profiles has been chosen.



The Slic3r tool change G-code provides the information for the wiping process. It is preconfigured and normally does not need interference.

Slic3r presets

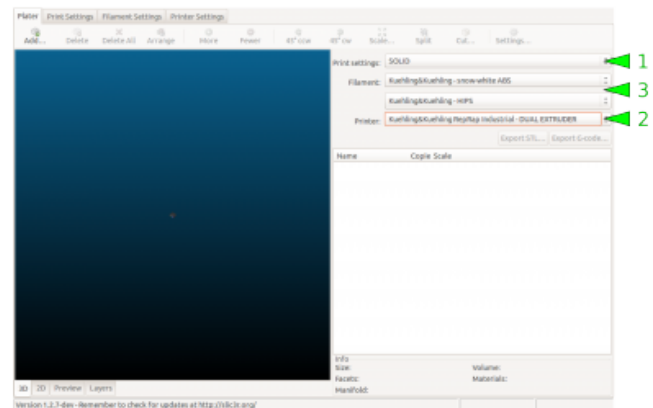
The full usability of the dual extrusion function is highly depending on the slicing software. As always, Kühling&Kühling recommend using [Slic3r](#) which exactly features this demand since the release of version 1.2.8. With the release of this version dual extrusion has become a more reliable, a more usable and a more useful feature especially when it comes to printing soluble support material. Download your [latest Slic3r version here](#).

The Slic3r-profile at our [GitHub](#) repository already come with tested and recommended presets for dual extrusion.

To begin with dual extruder printing the easiest way is to choose the respective preset in in the *Printer* menu of Slic3r:

1. Print settings

for the extrusion, there is no difference between single and dual extruder prints; choose SOLID, ECO or SOLID (soluble support) according to your needs. Regard that the extruder nozzles must be [assigned correctly](#).



Preselecting in Slic3r for dual extruder printing.

2. Printer

from the printer profiles select
DUAL EXTRUDER (bicolored,
nozzles 0.35+0.35)
or
DUAL EXTRUDER (soluble support,
nozzles 0.35+0.5) according to the
task.

After choosing either dual extruder profile
a second *Filament* dropdown menu
appears.

3. Filament

Choose the material type for each
extruder individually. The first (upper)
material is selected for the left extruder,
the second (lower) is selected for the right
extruder.

Regard that a second material can only be
chosen after dual extrusion has been
selected in the *Printer* dropdown menu.
Additional information on well-functioning
combinations of materials can be found in
the chapter [materials](#).

Upload your 3D model and adjust all necessary settings as usual.

For [supported prints](#) no further manipulation of the 3D model is required.

For [bicolored prints](#) additional preparations are necessary.

Printing with soluble support material

Common break-away support is not what takes 3D printing to its limits since its flaws nearly void the possibilities when it comes to surface quality. Due to the necessary gap between support structure and model underside, the later are always subject to gravity and will sag a little, reducing the contact to following layers. To really produce optically impeccable undersides a closely fitting, gapless support is required.

For printing with a soluble support material, it is mandatory that the slicing software provides the possibility of defining the distance between support and model. Only if no gap remains between these two, the superiority of soluble support compared to break-away structures becomes visible in the model.

The following Slic3r presets are provided with the Kühling&Kühling printing profile SOLID (soluble support). This description is meant to give you all necessary information to try out new settings.

Menu „Print Settings“

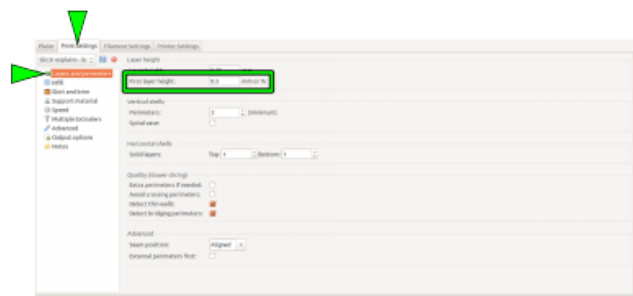
→ Layers and perimeters

- The First layer height must match the diameter of the support material extruder (the right extruder is preset).

INFO



This function cannot be used with any ECO profile or similar settings because the solvent for the support material may soak into the model which can cause long-term damage.



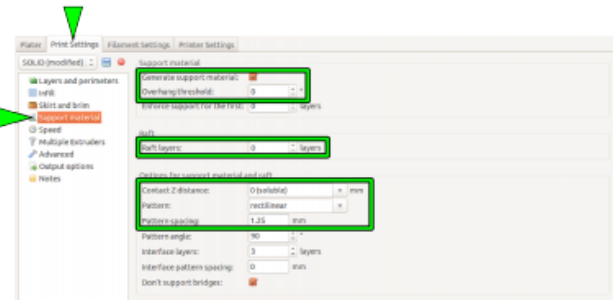
First layer height and support material extruder nozzle diameter must correspond.

→ Support material

- The checkbox **Generate support material** must be activated. If required,

the Overhang threshold angle can be adjusted accordingly (check via the *Preview* in the *Plater*).

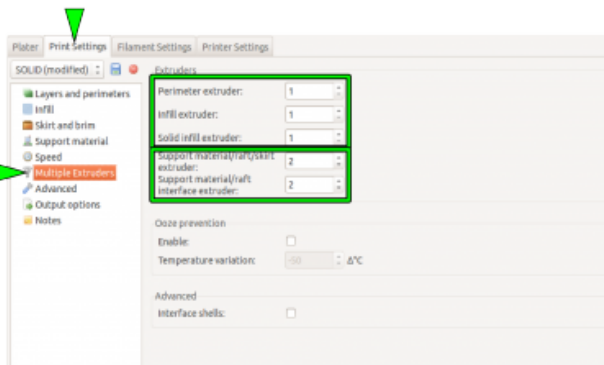
- The Contact Z distance must be „0 (soluble)“ with a (recommended) „rectilinear“ Pattern and a Pattern spacing of 1.25 mm.
- It is a good idea to generate 5 - 7 Raft layers with the support material. Thus, slight leveling mistakes are compensated and an even and reliable subsurface is always provided.



Choose these settings for dual extruder prints with soluble support material.

→ Multiple Extruders → Extruders

- Select the model and the support extruder according to the loaded materials.
Preset recommendation:
Perimeter/Infill/Solid infill extruder→Extruder 1
Support material/raft/skirt/interface extruder→Extruder 2.

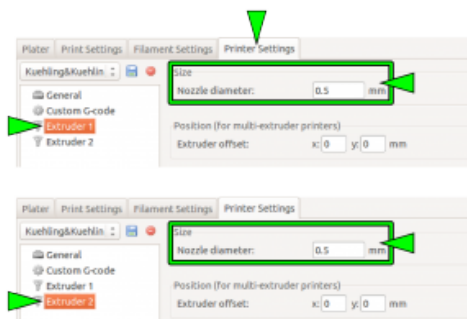


Assigning the hot ends to their respective tasks.

Menu „Printer Settings“

→ Extruder 1 / Extruder 2 → Size

- Check that the correct nozzle size is set for each extruder (remember the **First layer height**).



Make sure the correct nozzle size has been entered for each extruder.

Post-treatment of soluble HIPS support

ENVIRONMENTAL DANGER!

d-limonene is very toxic to aquatic organisms and must not enter the sanitation. Make sure to dispose of d-limonene in a suitable and environmental friendly way. Regard the manufacturer's safety data sheet when handling d-limonene.

FIRE HAZARD!

d-limonene and its fumes are flammable.

Do not handle in the vicinity of heat, open flames, hot surfaces and other ignition sources.

Do not smoke.

Regard the manufacturer's safety data sheet when handling d-limonene.

DANGER OF INJURY!

d-limonene is a solvating agent and therefore must not be handled carelessly.

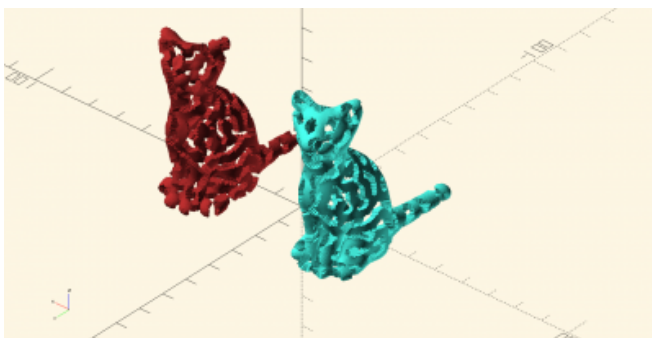
d-limonene can cause skin and eye irritations.

Wear suitable protective gloves and goggles and regard the manufacturer's safety data sheet when handling d-limonene.

After a complex ABS model has successfully been printed with HIPS soluble support material, the support material must be removed and dissolved. This is an easy process that only requires only a few steps:

- Mechanically break away the easily accessible parts of the support material.
- Place the model in a bath of [Kühling&Kühling wash-out solution](#) (97% d-limonene - order directly via sales@kuehlingkuehling.de) so that it is fully submerged. The bath is best stirred, either with a magnetic stirrer or by other means to provide its full solvating capacity and to reduce bathing time.
- When all remaining HIPS has been dissolved (check occasionally) remove the model from the bath, briefly dry it with a paper towel, and leave it in a warm and bright place (e.g. windowsill) to dry completely.

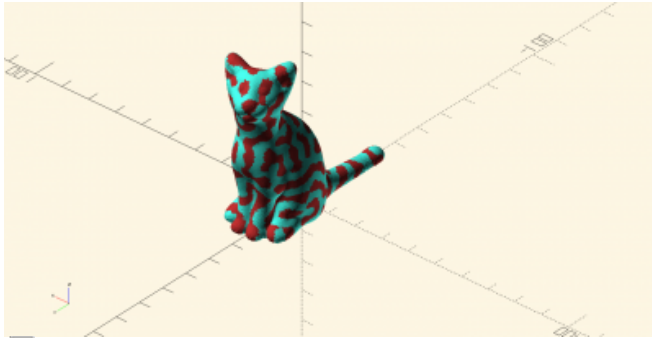
Bicolored printing



3D model preparation for bicolored printing

example 1: a model

(<http://www.thingiverse.com/thing:62536>) split in two for bicolored printing. Both must be aligned with the same relative position and exported as separate STLs.



The two parts correctly positioned and ready for export.



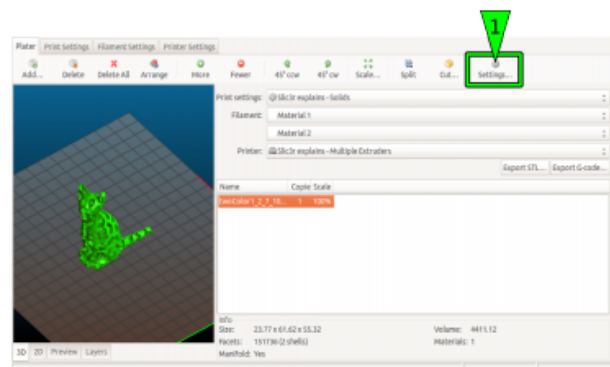
3D model preparation for bicolored printing
example 2: both individual parts of the 3D model being aligned in a common assembly and exported as separate STL-files with the correct relative coordinates.

The model preparation for bicolored (and also multi-material) prints is somewhat more complex than supported prints since Slic3r cannot distinguish single areas of a certain model and therefore is not able to automatically calculate the necessary commands. The respective 3D model must be designed as two separate parts with exact relative coordinates to enable Slic3r positioning them as required.

It is best to create the model from two separate parts and join and position them in a common assembly before exporting each as STL, so that the relative positions are recorded in the individual files. Slic3r has the feature to combine multiple STL files into a multi-material print job. Split the original design into the separate parts within the CAD program, and export each part as STL. Make sure both parts do not intersect.

Uploading and slicing the parts:

- Upload the first part to the *Plater*.
- Open the Settings dialog.

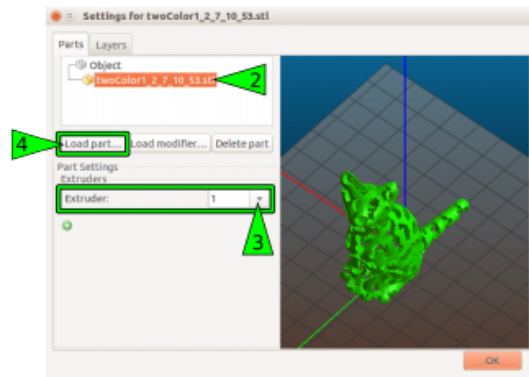


The first part is loaded to the Plater directly.

- Mark part one and assign it to extruder 1.
- Click Load part ... and add the second

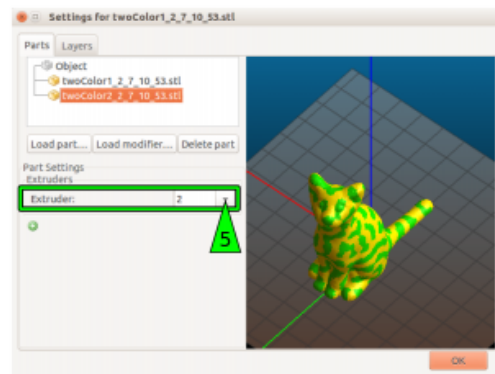
part from your file system.

If the single parts are aligned correctly, the preview should now show the exact representation of your desired model.



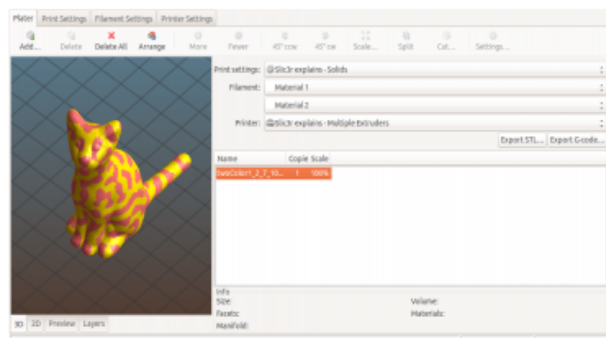
After the extruder for the first part has been assigned, the second part is loaded via the *Settings* dialog.

- Mark the second part and assign to extruder 2.
- Close the settings by clicking OK.



The second part must be assigned to extruder 2.

- You can add and load more than one part and modify it as described above.
- Adjust all other necessary settings as usual and as described in [Basic dual extruder settings](#).
- Prepare the 3D printer as described in [3D printer preparation](#).



This is what the preview should look like: both parts clearly visible in their correct position.



You can fill the print bed with different objects.

Print preparations

The term *print preparation* summarizes information about:

- Determining material properties
 - [Material specific extrusion temperature](#)
 - [Finding the extrusion multiplier](#)
 - [Filament quality](#)
- 3D-suitable construction and slicing
 - [Handling 3D-files](#)
 - [Layers and quality](#)
 - [Dimensional accuracy](#)

and the like, which may be useful or required prior to printing.

Setting the extrusion temperature

Depending on the material and even with the same but differently colored material the extrusion temperature varies strongly. Most manufacturers provide suggestions for their materials which are always a good idea to start with when printing a new material for the first time.

Fine-tuning the temperature can improve results or reduce unwanted side effects (such as oozing and stringing). Some materials react astonishingly different within a temperature range of ± 5 °C. For this reason, beginning with software version v1.1.0, the GUI of the HT500 is equipped with a direct temperature control and extrude function for quick-and-easy evaluating different temperature settings.

This new feature was installed to give you the opportunity of finding the correct extrusion temperature of any material more directly and comfortably without having to use the [Prime Extruders] wizard.

To find a suitable extrusion temperature:

1. Open the *Expert Control* menu.
On the right hand side of the screen you will find buttons to choose the extruder (left or right).
2. Choose an extruder.



Try finding the suitable temperature by directly extruding via the [Expert Control](#) screen.

INFO



Observe that the temperature shown in the footer of the GUI is measured at the hot end heater and may be 5 - 10 °C lower at the nozzle tip. Try increasing the temperature if you are discontented by the printing results.

After selecting ([Left]/[Right]) the menu is extended. The starting temperature is displayed according to the selected filament profile (see [Configuration](#)).

3. Enter the extrusion temperature recommended by the manufacturer via the touchscreen keyboard. The temperature can be increased or decreased in steps of 10 °C (tap [++] / [--] once) or 1 °C (tap [+] / [-] once).
4. Tap [set] when the desired value is set. The 3D printer will heat to this temperature and then unlock the [extrude] and [retract] buttons.
5. Tap [extrude] to extrude 5 mm of filament (tapping repeatedly will extrude 5 mm per tap). Tapping [retract] pulls the filament 5 mm back for every tap.
6. Watch the extrusion and check whether the following recommendations are visible:
 1. The filament is extruded smoothly in a single, clean strand.
 2. No bubbling or discoloration is visible. Bubbling or discoloration mean an **extrusion temperature set too high**.
If bubbling does not cease after reducing the temperature, this may also mean that the material absorbed moisture. It must then be dried thoroughly.
 3. The extruded strand is not swelling up after leaving the nozzle tip. Upswell means the **temperature is too low** and the internal friction of the material too high.
 4. No slipping of the extruder drive gear is visible. Slipping of the drive gear indicates an **extrusion temperature set too low** and a too high frictional resistance in the nozzle.
 5. The material is not oozing excessively from the nozzle tip after finishing the extrusion and reducing the temperature.

If these points are not met, adjust

When extruding new materials do not forget to print a [calibration cube](#) to find the correct extrusion multiplier.



When no filament profile is active for the extruder, the starting temperature is set as 180 °C.

INFO

„Normal“ oozing is differing largely between materials. Some materials (like ABS) are highly viscous at extrusion temperature and oozing only a little, other materials (e.g. Nylon) are melting in a wide temperature range and are flowing after until the nozzle tip is empty due to their low viscosity. Finding the best extrusion temperature requires some experimenting. Generally it is a good idea to start with the temperature recommended by the manufacturer and steps of 5 °C above and below until the individual requirements are met.



the temperature and extrude again until the correct appearance is visible.

7. Tap [OFF] to deactivate the extruder heating unless you want to keep the current extruder temperature for further steps.
8. To verify the settings, print a sample model and check the extrusion properties. Adjust the settings if required.

But as much as correct temperature settings the G-code (i.e. the Slic3r settings) influences the print process and the final result. Make sure to use the software and hardware settings suitable for the print job. There is no need to print with an 0.25 mm nozzle tip at low speed when printing large, straight-edged objects.

We ourselves are constantly testing new procedures, new materials and new geometries and anything we find out will be shared in the documentation.

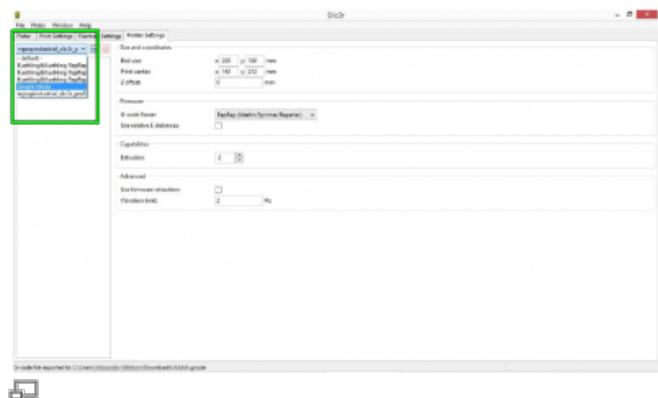
Handling 3D files

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.



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

Try the following to bypass the problem:

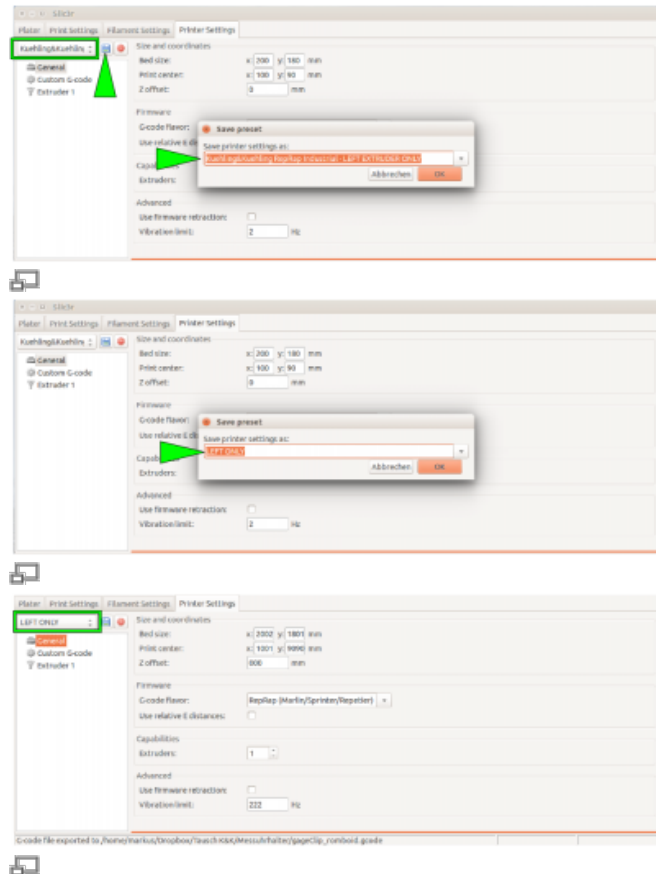
- Select a profile from the drop-down list and click the <save> symbol.

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 HT500 - LEFT EXTRUDER ONLY“.

- The shorter description will be fully visible in the list.

Repeat these steps for all long profile names.



Easy evaluation of STL-files

Especially when using files downloaded from open internet sources it may be that these files are not suitable for printing due to export or modeling mistakes. Misaligned edges (non-manifolds) and holes can render a 3D-model unprintable and make troubleshooting on the 3D-printer an infinite displeasure because no obvious reason can be detected.

One quick and simple way to eliminate this as an error source is checking the suitability of your STL-file for 3D-printing. All you need is a program capable of analyzing the STL-file's mesh.

Slic3r itself can deal with minor troubles and the [netfabb company](#) offers a [free basic version](#) of their *netfabb studio*, an STL-handling tool with repair functions. Other tools may be as helpful but these two are used to explain how to recognize a corrupted file.

Slic3r part info:

(see [Slic3r manual](#) also)

After opening a file in Slic3r, the info box in the bottom-right corner of the plater shows information on the model status. For a single, suitable part the info must look like:

Factes: xxxxx (1 shells)

Manifold: Yes

If it displays Manifold: Auto-repaired (xxxx errors) the file contains degenerations.

It is most likely that it also shows an increased

Info		
Size:	40.94 x 40.94 x 20.00	Volume: 15447.04
Facets:	4180 (1 shells)	Materials: 1
Manifold:	Yes	

Info		
Size:	19.16 x 4.80 x 23.04	Volume: 309.72
Facets:	22462 (6 shells)	Materials: 1
Manifold:	⚠Auto-repaired (7399 errors)	

Slic3r info box for a flawless (above) and a corrupted (below) part.

number of shells (e.g. Facets: xxxxx (6 shells) because the part has been split. Also check the 3D-preview for visible defects. In this case it is better to repair or reconstruct the model. Regard that the auto-repair function of Sic3r is only suited for repairing minor defects.

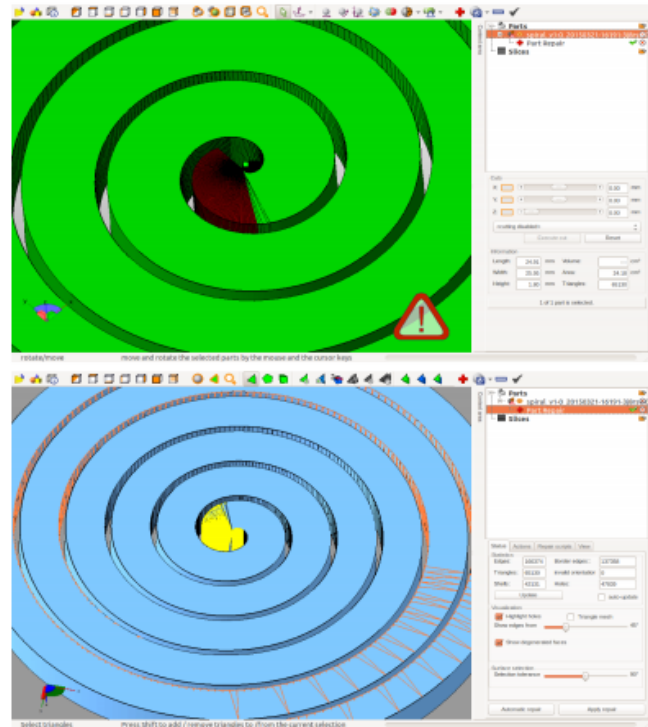
Evaluation and repair with netfabb basic:

(see [netfabb basic manual](#) also)

If you open a (supposedly broken) STL in netfabb, the program will display a red warning sign in the bottom-right corner when the part is indeed faulty.

To see mismatches, run the part repair function by clicking on the red cross sign in the toolbar or via the *Extras* menu. The part is analyzed and all mesh information are displayed, including holes, invalid orientation etc. You can try using the auto-repair function or manually repair defects (very limited in the basic edition) to correct the part prior to newly exporting it as STL.

If simple repair does not lead to satisfactory results, redesigning or renouncing this specific part and choosing another one may be the best choices.



Alert of mismatches in an STL and the preview of misaligned edges and holes in netfabb basic's part repair.

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.

Modifying infill and perimeters

There are currently two Slic3r profiles available at our [GitHub repository](#) that provide preset ready-to-print slicing settings. Here 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.

Layers and quality

If it comes to high-quality printing, there is no way to avoid grappling with the „layers“ topic. The layer settings influence the quality of a print in the same way as extrusion temperature and movement speed do. In fact, all of these settings interact and influence one another, but you may have recognized that in every slicing software there are a lot of layer-specific settings waiting for adjustment.

The following paragraphs are meant to give you an overview on the topic and further to provide you with detailed information about necessary basic and useful advanced settings to stepwise increase your prints with the HT500.

As previously mentioned elsewhere in this manual, we are focused on printing ABS at high quality so the following mostly refers to settings with this material; if otherwise, it will be specially emphasized. When talking about software settings we always refer to the Slic3r software of Alessandro Ranellucci and his team (<http://slic3r.org/>).

Everything else only refers to printing with the HT500, since this is the basis of our experience. Most things may also be valid for other appliances and different slicing software since it is basic knowledge about FFF manufacturing which can be found elsewhere in books and on the internet. We want to enable the user of a HT500 to achieve the best possible results with his machine and to provide the necessary knowledge from a single source.

Warping prevention

One of the most important factors deciding whether a print finishes at all and if it does defining the final quality is the adhesion of the first layer to the print bed. The first layer channels off heat tensions from the printed object into the print bed.

ABS, Polycarbonate and Nylon for example are very prone to warping, so only firm adherence to the subsurface can effectively prevent bending and warping. With a first layer not sticking to the subsurface all-over, tensions will manifest all through the model - the object will detach itself from the print bed and the print is wasted. High temperature effected tensions manifest ever stronger with increasing height of the printed model. Long, straight perimeters enhance this effect further.

Three factors determine the adhesion:

1. A correctly leveled print bed (explained under [Tips & Tricks](#)),
2. the material of the print bed and
3. the temperature of the print bed.

The material of the print bed must match the chosen filament. The standardized PEI print beds delivered with the HT500 perfectly match the requirements for printing [ABS](#) and [HIPS](#), the most common printing materials for this 3D printer. Other materials such as [PET Copolyester](#) and [TPE-u](#) also stick to the PEI surface unproblematically.

If a special treatment of the print bed (e.g. PVA glue for printing [PVA](#)) or another material (e.g. plywood for printing [PA 12](#)) are required, this is specially stated in the list of [Materials](#).

The most necessary print setting apart from the material is the correct print bed temperature to ensure that the filament sticks to its subsurface throughout the printing process. Some materials, like ABS or PC, need high temperatures to stick, whereas for others, such as Nylon or PVA, heat is counter-productive and a cold bed is needed.

The required print bed temperature for any material tested on the HT500 3D printer can also be found in the [Materials'](#) descriptions.

Other Slic3r settings that help minimizing warping are described in the [Tips & Tricks](#) rubric.

The first layer

One thing that always stands out when preparing a print job with the slicing software are the **first layer** settings. While everything thereafter is only „layers“, the first layer is especially important for the result of any print job.

This is because of the requirements imposed on the first layer:

- Compensation of unevennesses of the subsurface to found an even foundation for the following layers. Slight leveling mistakes, scratches and bends can provide quite an uneven underground; in the slicing profiles provided at our [GitHub repository](#) the first layer is preset to be thicker and wider (see [below](#)) than the following layers. The increased throughput during printing the first layer fills in waviness of the subsurfaces and thus provides an even underground for the following layers.
- Sticking to the print bed during the entire print job so that heat tensions are channeled off and warping is minimized. The adherence to the print bed is above all responsible for the minimization or manifestation of warpage. Correct first layer settings become the more important the larger the printed object is and the more long, strung-out edges it contains.
- Loosing the adhesion after completion of the print job. After finishing a print job, the model must be detached from the print bed without breaking apart so the first layer must not merge permanently with the print bed but reversible (e.g. due to cooling). This is the only feature that cannot be influenced via the slicing settings and is totally dependent on the material match of printed plastic and print bed.

It has been a challenge to find materials matching the properties of ABS and the above named special features. After a lot of testing, we found that our PEI print bed meets all requirements concerning rigidity, stiffness, flatness, and thermal stability. In combination with the heating of the build chamber we experience no trouble printing straight, non-deforming objects and have had no problems so far removing our models from the print bed.

If you are sure to have leveled your print bed correctly and all temperatures are set right and still you experience delamination and warpage, have a look at your first layer settings; there might be something to be improved here.

Mechanical features for a high-quality first layer

Since the first layer is a well known problem of 3D printing and distinguishes the over-all quality of a 3D Printer, Kühlung&Kühlung focused on providing a strong mechanical basis from the beginning on.

To make sure that constructive negligence never may ruin a print, intense attention was given to the rigidity of the print table, the choice of an adequate print bed and an easy-to-use and reliable design of the leveling mechanism.

The print table is designed to be stiff and rigid with a precisely manufactured aluminum plate as subsurface for the print bed. The crossbeams have been kept as short as possible to eliminate leverage and the carriage is mounted on high-precision shafts and guided by minimal friction linear slide bearings. The drive spindle allows for extremely precise positioning of the print table in Z-direction with neglectable gravity-compensated backlash.

With the newly designed silicone heating pad of hardware revision 1.1.0, the heat distribution has been improved to be still more even to ensure an over-all heating of the print bed.

The print bed itself is a particularly chosen and tested PEI-glassfibre plate that provides especially good properties for printing ABS. It perfectly spreads the heat provided by the silicone heating pad and, though not thicker than 2 mm, is rigid enough to withstand warping forces as well as sufficiently flexible to remove finished print objects by bending and flexing.

But no constructive measures will suffice if the print bed is askew, so precise leveling is mandatory. To keep the leveling process as time saving as possible, the print table is mounted on a three-point spring support with set screws to fixate it in the correct position. This guarantees the necessary uniform distance of nozzle tip and print surface all over the print bed. The respective leveling instructions are implemented on the touchscreen user interface as a step-by-step wizard that is simple to follow and quick to perform.

First layer settings

As mentioned above, the following recommendations are valid for printing ABS on the HT500. We have taken all this into respect when creating our slicing profiles and you do not need to try and find matching settings for the first prints. After growing accustomed to 3D printing and the apparatus' specific features you can start optimizing. Also, if you want to print other materials than ABS, the basic settings named here apply and can be used to modify your results.

The first layer should be thicker and wider than the following layers. In case of the HT500 we recommend the following (and have already taken it into respect in our slicing profiles):

- The first layer height should be adequate to the nozzle tip's diameter.
For example:

INFO

The following applies to the nozzle tip size 0.35 mm and the preset Slic3r profiles you installed during initial commissioning. It is meant to give you an overview of the settings relevant for the optimization of the first layer adhesion. Please experiment to find the settings best suited for your task and test all settings before applying them to your final model if you use other nozzle sizes.

When using the 0.5 or the 0.75 mm nozzle tip, there is no further improvement when setting the layer height equal to the nozzle tip diameter. A value of 0.4 mm will suffice for these nozzle tips.



Invalid Link



Slic3r Print Settings - setting the first layer height

Printing with a nozzle tip size 0.35 means setting the first layer height to 0.35 mm

- The extrusion width should be set to 150 % of the first layer height.
For example:
Printing with a nozzle tip size 0.35 and thus 0.35 mm first layer height, setting the first layer extrusion width to 150 % means an extrusion width of 0.53 mm. This ensures that enough filament is extruded to compensate for slight unevenness of the subsurface and still enough adhesion. In Slic3r you can preset and save this setting as a percentage so that the first layer's extrusion width is independent of the actual nozzle tip size.

Invalid Link



Slic3r Advanced Settings - reducing the first layer extrusion width

INFO



Both above mentioned measures compensate for unevennesses of the print bed and increase the tolerance against slight leveling mistakes. The distance between nozzle tip and print bed will assure that the bore is not clogged when passing convex bumps by leaving enough free space in between. The extrusion width will make sure that enough material is conveyed to equalize concave crudities.

- The first layer speed should be reduced because the print bed does not fuse easily with the material. Slower movement ensures a more even temperature distribution between underground and material and thereby a better binding between them as well as longer cooling time which aids relaxation of tensions. For ABS, a first layer speed of 15 mm/s is a functional value. For other materials, settings from 10 to 30 mm/s are recommended (always choose slower values for nozzle tips ≤ 0.35 mm).

Invalid Link



Slic3r Print Settings - setting the first layer extrusion speed

- [Changing the extrusion temperature](#) and the print bed temperature can also improve the first layer adherence. Changing the temperature a few degrees may allow for the filament to melt a little more inviscid and to lay more evenly onto the print bed. Increasing the print bed temperature can have a similar effect as reducing the extrusion speed: a better temperature distribution and increased cooling time. For printing ABS we find no difference in

Invalid Link



Slic3r Filament Settings - setting the first layer extrusion temperature

INFO



*Optimize the extrusion and print bed temperature for the [Material](#) currently installed.
Do not hesitate to try different settings here, since even colored*

quality using varying temperatures so the settings are the same for the first and the following layers but this feature may help printing other plastics.



additives may alter the optimal extrusion temperature.

- For filigree or thin-walled objects add a 3 to 6 mm brim and/or 3 to 8 raft layers to increase the effective adhering area (see [Slic3r manual](#) also).
- Adding a raft may also compensate leveling mistakes as described [here](#).

Invalid Link



Slic3r Print Settings - adding a brim

Invalid Link



Slic3r Print Settings - adding raft layers

These are the settings for the first layer. Try and experiment with them to improve your results or to print new materials. We shall do the same and publish all relevant findings.

Adhesion faults

The fault easiest to diagnose and remedy is the presence of adhesives on the print bed. Recently removed an object from the print bed? Maybe touched the surface? Fingerprints are very effective separating agents that will with certainty prevent sticking of the filament. Any oil, fat, grease or the like will have this effect, so make sure you always [clean](#) the print bed with a suitable solvent before starting a new print.

In most cases, [false leveling](#) of the print bed is the reason for unsatisfactorily adhesion of the first layer. If the distance between the print bed surface and the nozzle tip becomes too wide, the filament will not be pressed but laid on the surface and single strands will not merge. No cohesion between adjoined strands and no adherence to the underground will develop. It is easy to recognize and as easy to correct such faults as result from false leveling.

Too low a temperature of the print bed also effects poor first layer adhesion. In such cases it might be necessary to find the correct temperature by simply trying. If you cannot succeed with this, something might be wrong with the heating element.

Following layers

Whereas the first layer decides if a print can be finished or not, the following layers influence the fineness and optical appearance of the model. Thinner layers improve the surface quality and chamfers and curvatures are more refined and smooth with much less visible steps. But choosing a lower layer height will also increase the printing time. Regard that halving the layer height more than doubles the print time and at some point the improvement of the appearance will be no longer visible. The other way around you will receive a rougher surface in a shorter time. It is depending on your demands which resolution you try to achieve and how much time you are able or willing to invest for a certain result. The slicing settings are not limited in this regard, so feel free to try it out. We recommend the following settings when printing ABS or other viscous materials:

- The minimal layer height should not undercut 0.1 mm. For values below this we have no adequate experience since the improvement in surface resolution stands way behind the increase of print time. More inviscid materials may allow even thinner layers. Regard that the layer height is not necessarily limited by the nozzle tip diameter; it is possible

to receive excellent results printing 0.1 mm layers with a 0.35 mm nozzle tip. In the same regard take note that the layer height does not implicitly improve every result. When it comes to fine structures and sharp edges, choosing a smaller nozzle tip without reducing the corresponding layer height can be the key to success.

- The maximal layer height should not exceed 80 % of the nozzle tip diameter. Above this value the distance between nozzle tip and previous layer becomes too far and the extruded amount of material may not suffice for good layer binding and thus strong, stable objects. Slicing software like Slic3r do allow configuring layer heights up to the actual nozzle tip diameter (100%) but broad experience in FFF 3D printing and the Open Source RepRap 3D printer development shows, that max. 80% is a safe approximate value for best results.

Dimensional accuracy

Dimensional accuracy is a core demand of any manufacturing process and therefore is a feature very much looked into in the HT500 3D printer. It is built to provide high mechanical accuracy in positioning and supports a variety of possibilities to compensate for mechanical influence factors to improve appearance and dimensional accuracy (e.g. precise [backlash calibration](#)).

Also, the recommended slicing software [Slic3r](#) calculates extrusion width, material flow and movement to precisely match the dimensions given in the .stl-file.

Kühling&Kühling aim to meet accuracy demands of ± 0.2 mm „out-of-the-box“, thus delivering every HT500 3D printer carefully adjusted and optimally calibrated.

But some factors that influence the dimensional accuracy of print objects cannot be compensated by the apparatus or the slicing software but can be partly neutralized by slightly altering the model geometry of the CAD-file.

Currently, it is not possible to compensate all influences equally and at the same time but the following tips provide some means to improve the dimensional accuracy of your prints if required.

Influence of thermal expansion

Any material is subject to thermal expansion when heated and shrinkage when cooled. Since the temperature range a plastic passes through during a print is very wide, reaching from around 100 °C at the print bed surface to approx. 20 °C when taken out of the build chamber, the influence of thermal expansion cannot be neglected. Thermoplastics usually have a relatively high thermal expansion factor.

For example: the thermal linear expansion coefficient α_L of ABS is approximately $1 \times 10^{-4} \text{ K}^{-1}$, which means that a part of 25 mm length (or diameter) will shrink by approximately:

$$25 \text{ mm} \times 1 \times 10^{-4} \text{ K}^{-1} \times (100 \text{ °C} - 20 \text{ °C}) = 0.19 \text{ mm} (\cong 0.76 \text{ \%})$$

due to thermal influences alone.

A series of over two hundred measurements with ABS test bodies printed with a HT500 has shown that the effective deviation lies above the expected value, between 1 and 2%, increasing with decreasing size. Especially for small dimensions ≤ 5 mm other effects (see below) may counter or intensify the deviation.

For nominal values between 2.5 and 25 mm, the deviation was measured for uncompensated test bodies and two different methods of compensation.

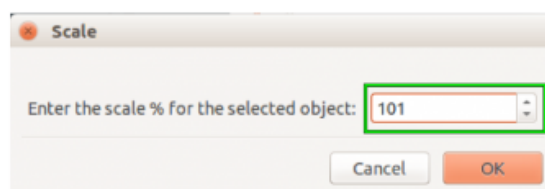
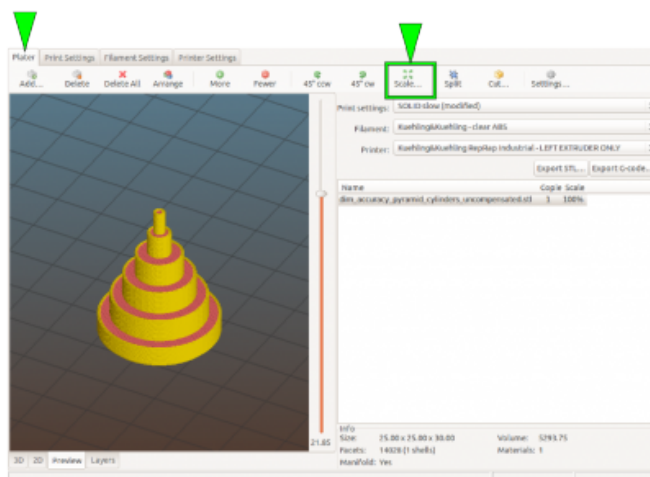
For the uncompensated measurements the above said applies.

The first and more complicated method of compensation is to directly apply the thermal expansion factor in the 3D CAD model and producing an .stl-file that already comes with corrected dimensions. This is only applicable if the 3D CAD program you are using provides the possibility to multiply all dimensions with a specific factor at once. This method is advantageous if you want to check dimensions in the CAD-file, which is much more common for .stl-data than ready-sliced g-codes.

The second, more comfortable way is to directly upscale the model in Slic3r.
For that:

- Upload your .stl-file to Slic3r.
- Choose the [Scale...] button and enter the percental scaling factor.

A factor of 101 % has proven to reduce thermal induced deviation to values between 0.15 and 0.75 % for values ≥ 5 mm.



 Scaling the model directly in Slic3r.

Excerpt from the measurement report:

Nominal value	Mean value (12 measurements)	Deviation	Compensated via 3D CAD model (dim x α_t)	Deviation	Compensated via Slic3r (101%)	Deviation
[mm]	[mm]	[%]	[mm]	[%]	[mm]	[%]
25.00	24.74	1.04	25.05	0.19	24.99	0.03
20.00	19.77	1.17	19.96	0.20	19.97	0.16
15.00	14.75	1.64	14.91	0.62	14.89	0.71
10.00	9.84	1.65	10.03	0.30	9.97	0.27
5.00	4.94	1.18	4.99	0.30	4.98	0.50
2.50	2.40	4.20	2.44	2.40	2.45	1.93

Illustration for Diameters

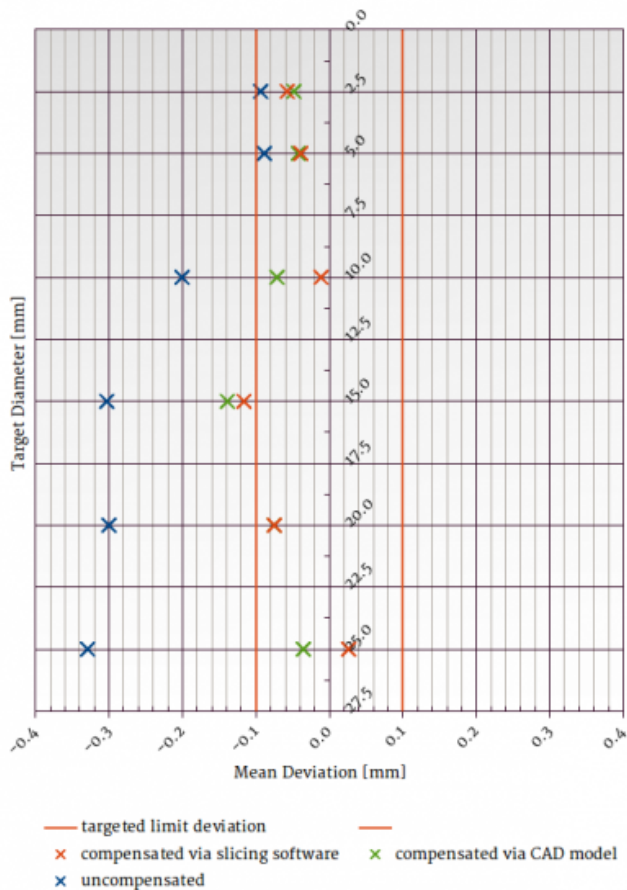
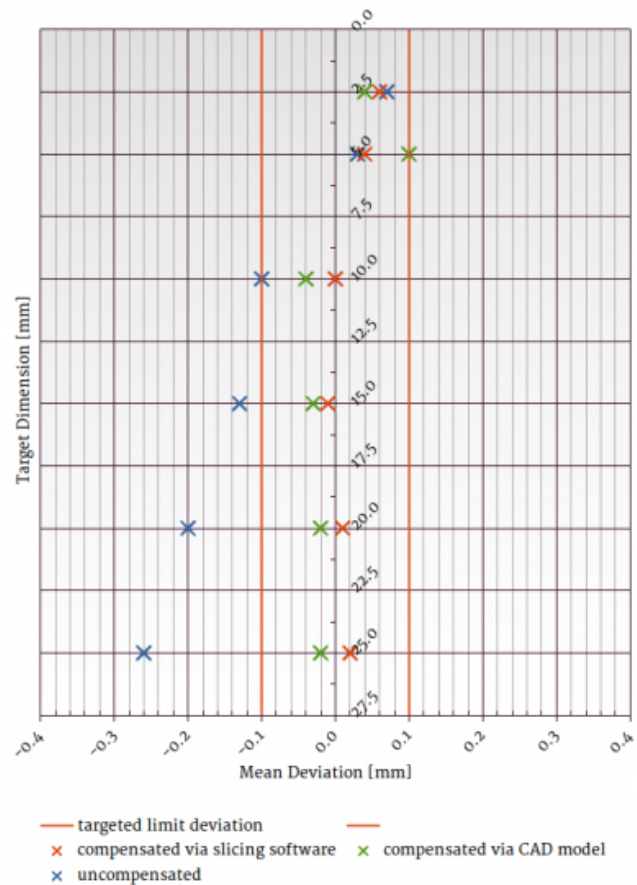


Illustration for Linear Dimensions

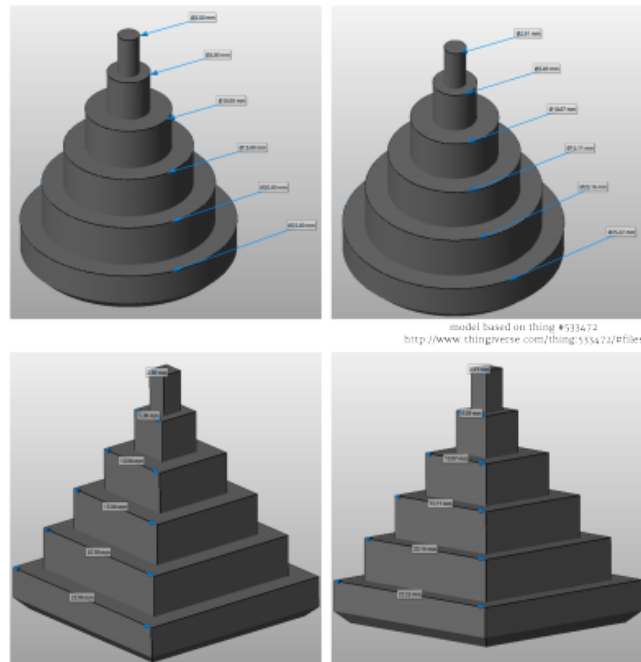


The deviation decreases significantly when the thermal expansion is compensated. Either for diameters of circular structures ...



... or linear dimensions.

The table above outlines that the improvement by either method is significant compared to the uncompensated model while not making much of a difference compared among each other. All measurements were done for a set of twelve individually printed test bodies (see adjacent picture). The deviations of each set and diameter were averaged for comparison. The table also shows that for smallest structures the effect is in fact significant but that other effects need compensating measures too to receive best results.

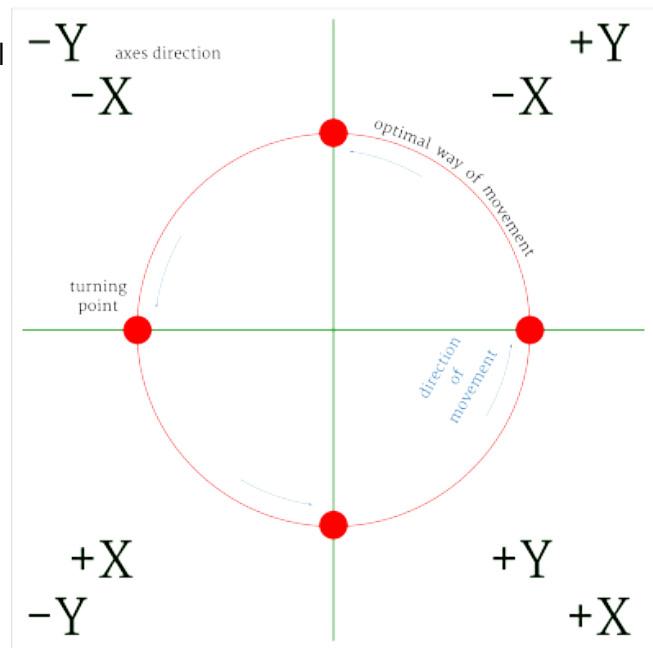


netfabb visualization of the ABS test bodies printed for comparison and evaluation. On the left hand-side the uncompensated model, on the right-hand side the model after multiplying with

α_L prior to exporting the .stl.

Influence of backlash

Another factor strongly affecting the dimensional accuracy is the driving unit's backlash, a mechanical characteristic of any device using drive gears, belts and spindle drives to create movement. Whenever the major direction of movements of one of the axes changes and the drive reverses direction, mechanical lash effects a slight offset from the optimal way of movement. This offset, the „backlash“, can be measured and compensated by software means to correct the way of movement whenever necessary.



The four directions of movement and the turning points during printing a circle.

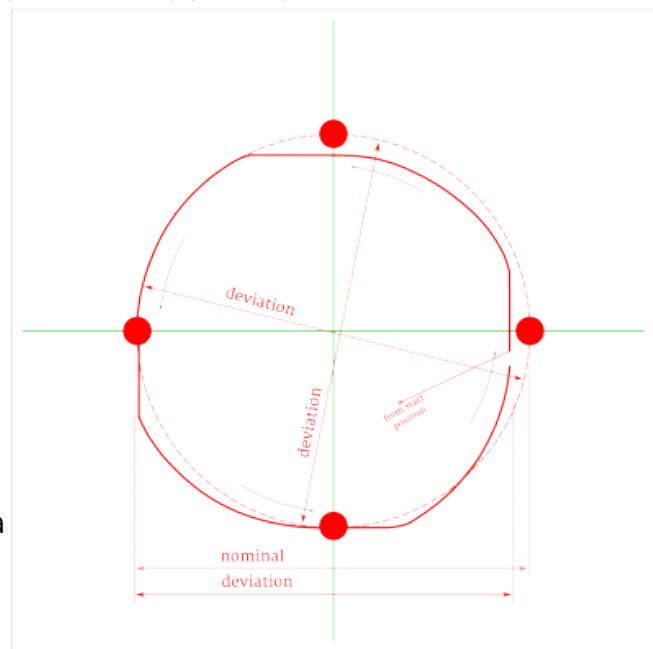
The adjacent pictures exemplarily show the influence of not or incorrectly compensated backlash on the dimensions of a circle, because this is where it is most prominently visible since the direction changes four times.

These figures are based upon a computed simulation with equal settings for the X- and Y-backlash compensation.

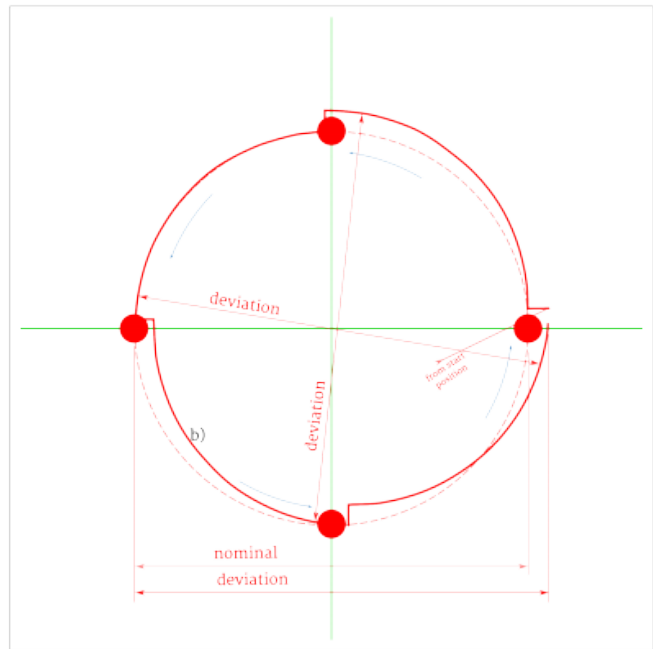
For either movement it is assumed that in the starting position (lower left) both belts are fully in the end stop of the pulley.

Regard that over-compensation does not effect a behavior similar to under-compensation, just on the other side, but creates a totally different movement pattern which is not predictable for more complex geometries.

The HT500 will show slightly different behavior since the axes' drives are differently arranged. The Y-axis almost always needs a higher compensation factor because of the superposition of the two drive belts. More machine-specific information on behavior and how to measure and compensate the backlash of the HT500 3D printer can be found in the [Service guide](#).



Effective travel of the print head when backlash is not or under-compensated.



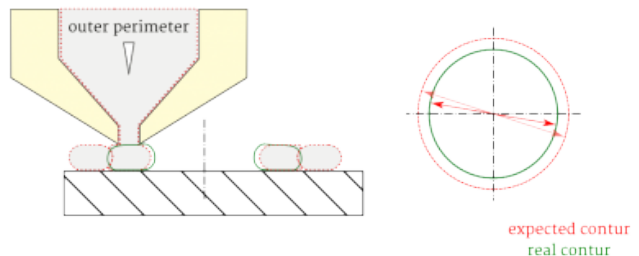
Effective travel of the print head when backlash is over-compensated.

Small holes

Small hole dimensions may be subject to two additional error causes apart from shrinkage and print head positioning.

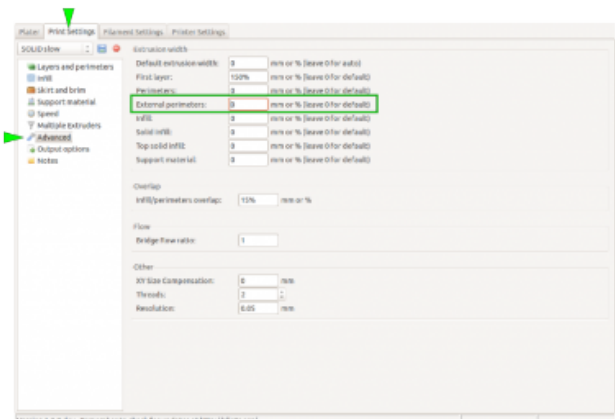
When the print head follows a curve the inner side of an extrusion strand is slightly overfilled which results in an excess of material on the inner wall of a hole.

The effect is strongest in holes of small diameters ($< \varnothing 5$ mm).



If your print object has little overhanging structures, reducing the outer perimeter extrusion width can help avoiding this effect (see graphic on the right). But most effectively it is countered directly in the CAD model since it is impossible to calculate and compensate by hardware and slicing software means.

Overfilling of the inside of the extrusion strand reduces the diameter of small holes.



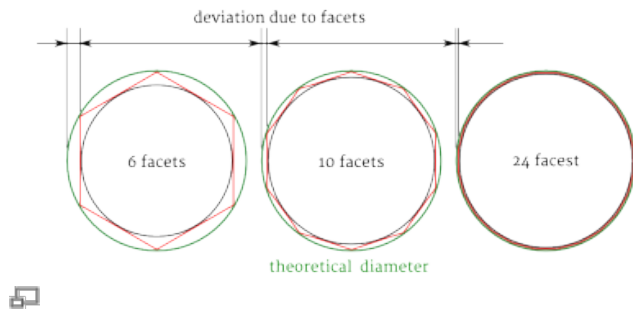
Reducing the outer perimeter extrusion width in Slic3r.

A minor aspect is the number of triangles a

model is built from, i.e. the number of facets. Normally, any .stl's resolution is fine enough for sufficiently reflecting the details of a model and respectively loss of accuracy is neglectable. Still, for small bores the number of facets may play a role if it is very low.

The graphic on the right displays the possible influence of facets on an inner diameter.

In case of roughly faceted objects try increasing the resolution of your CAD model to improve dimensional accuracy.



Correlation of low faceted walls and diameter.

Filament quality check

Much to our regret, filament manufacturing is not subject to standardization yet. Although we try to provide material of high quality and stable dimensions, manufacturing faults cannot be totally excluded.

If you order filament of other suppliers, please check back that it is within the stated tolerances. Some manufacturers offer a nominal „3 mm“ filament which really is 2.85 mm in diameter, while others indeed sell 3.00 mm.

If you note that material purchased via Kühling&Kühling does not meet the standards we set here, please let us know and we will promptly replace the material. We only ask you to send back the flawed material (return note will be included in the new delivery) so that we can discuss the matter with our supplier.

If you experience increased problems with grinding at the drive gear, first thing to check is the quality of the filament currently loaded:

- Is the diameter correct?
- Is the filament round?
- Is the filament free of bulges and kinks?

INFO

To make sure that there is no problem in the filament feed system, try the following:

- *Remove the filament feed hose from the filament inlet (push the blue fixing ring and pull the hose out).*

- *Feed the filament directly through the hose and try printing.*

If the problem remains, check the filament as described.

Refit the hose afterwards!



Diameter

Measure the diameter at several points (at least 5 per 0.5 m).

Measure at least twice (best thrice) at each point at an angle of 90° (60°). The diameter must not deviate from

2.85 ± 0.1 mm (2.75 ... 2.95 mm)

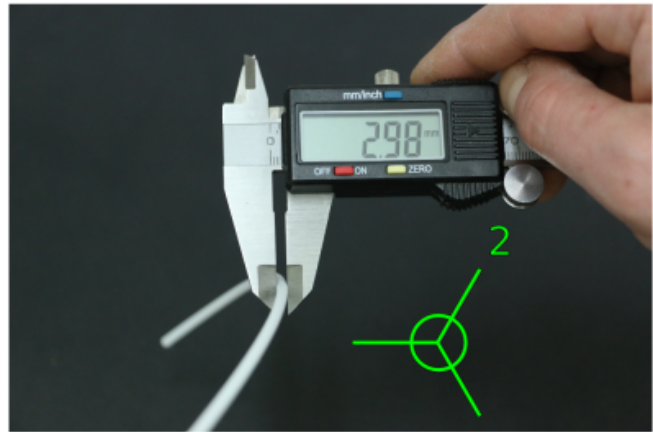
A diameter of e.g. 2.98 mm will increase friction in the filament feed system and the nozzle barrel and cause compression and jamming of the filament.

A diameter of 3.00 mm will no longer fit through the filament feed inlet.

Filament below 2.75 mm is, especially when a print job requires a lot of retraction movement, likely to jam the nozzle barrel because molten filament can push itself back up the barrel and form a plug.



First measured diameter.



Second measured diameter.



Third measured diameter - this filament exceeds the stated tolerance and must be replaced.

Roundness

Measure two times at the same point. If one diameter deviates from the stated tolerance or if both diameters differ strongly, the filament is out of round and must be replaced.

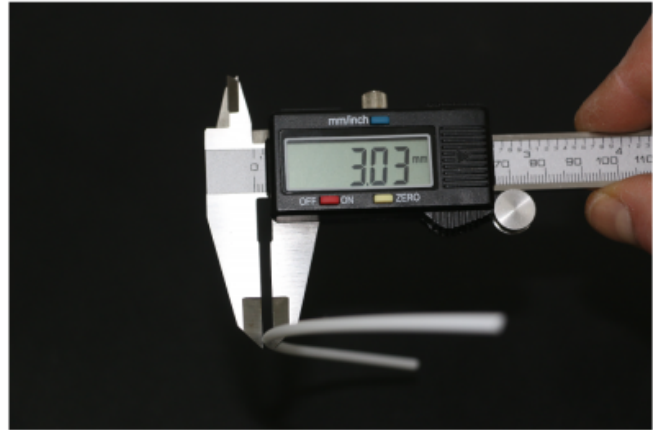
Out of round filament, even when meeting the admissible values, is likely to form a plug inside the nozzle barrel because molten filament can



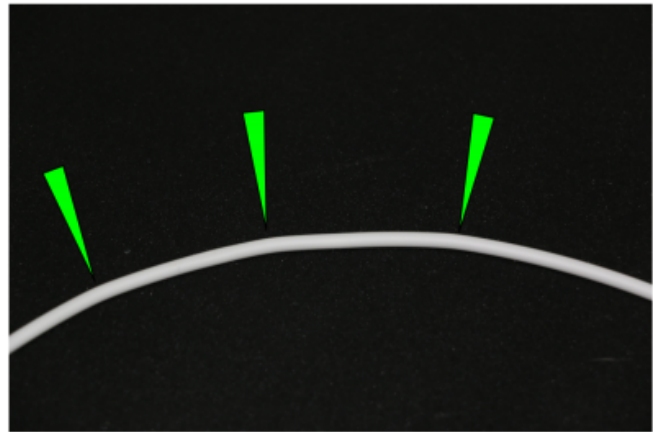
push itself back up past the main strand.



The measured diameter is below the admissible tolerance ...



Measured at the same position, turned 90°. The diameter now exceeds the tolerance - the filament is out of round and must be replaced.



Visible kinks in the filament may also lead to jamming.

Kinks and bulges

Visible quality failure such as kinks or bulges may also increase the friction in the filament feed system. Check for visible deformation.

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:

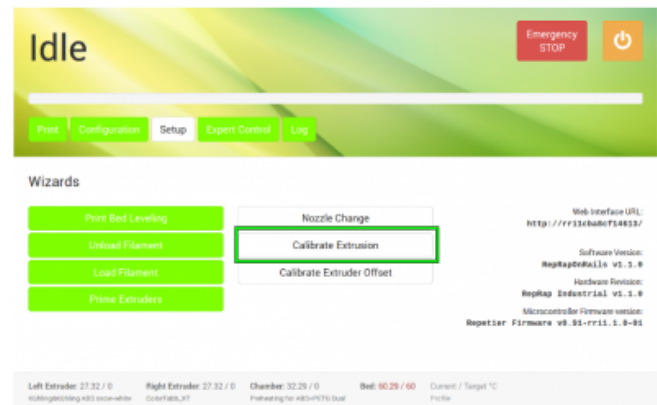
- the 3D printer itself - slight variations are possible due to the manufacturing process;
- the actual filament diameter - in the range of the dimensional stability of the filament;
- the printed material's properties - the extruder drive wheel grinds deeper into softer materials, thus reducing the 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.

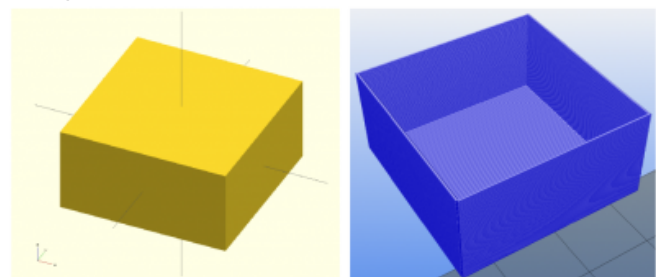
To find the correct multiplier, open the *Setup* menu at the GUI and choose [Calibrate Extrusion].

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.



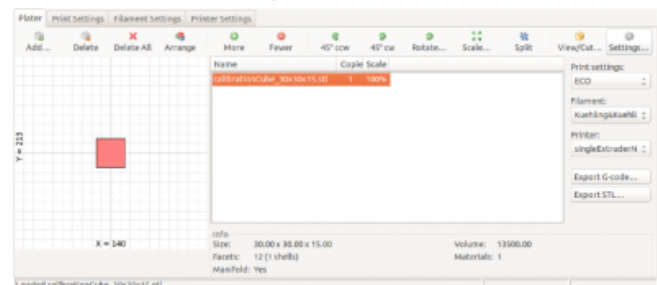
Choose the [Calibrate Extrusion] wizard from the *Setup* menu (either software version).

To create a G-code for the extrusion calibration, you need an stl-file of a cube with the dimensions 30x30x15 mm (download [here](#)).



The 30x30x15 mm calibration cube as OpenSCAD model and ready sliced with Slic3r (view taken with Repetier-Host).

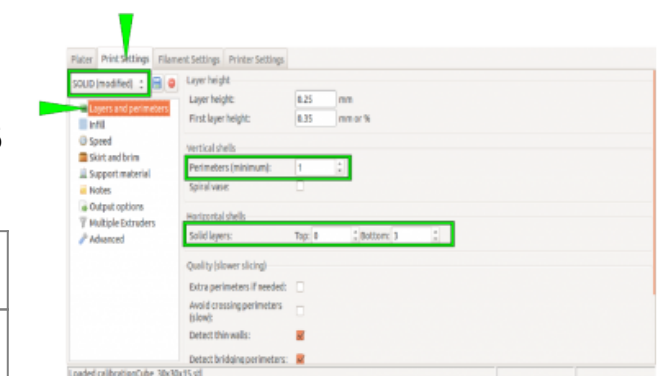
Load the stl-file in Slic3r and select the *Print Settings* tab.



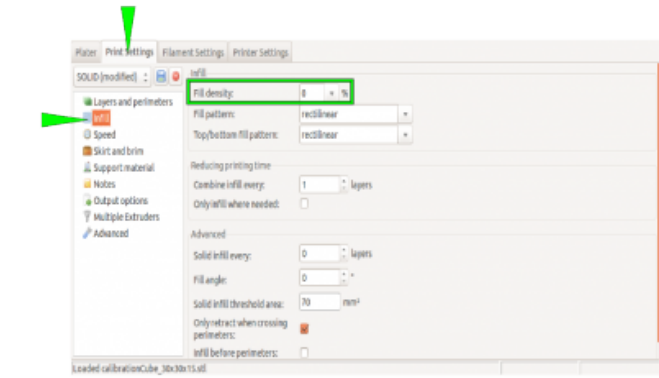
Load the calibration cube stl-file in Slic3r.

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:

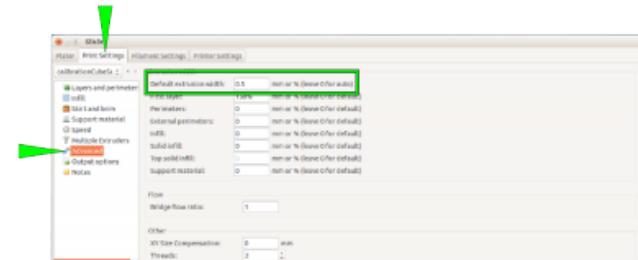
Layers and Perimeters → Vertical shells Perimeters (minimum)	1
Layers and Perimeters → Horizontal shells Solid Layers TOP	0
Layers and Perimeters → Horizontal shells Solid Layers BOTTOM	3
Infill → Fill density	0 %
Advanced → Default extrusion width	0.5



Select the *Layers and perimeters* according to the adjacent table.



Select the *Infill* according to the adjacent table.



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.

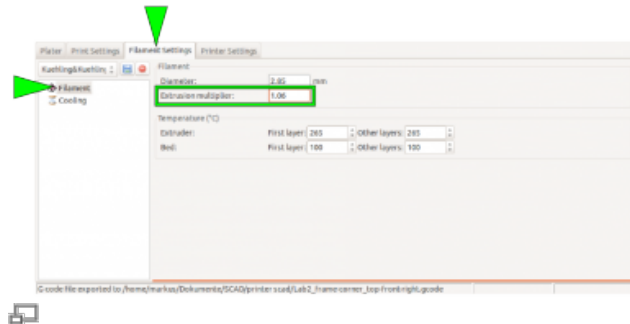
Upload the G-code to your 3D printer, print it, measure the wall thicknesses, and calculate the mean value.

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!).

Materials

All information given in the following have been thoroughly tested on the HT500 and are valid for this 3D-printer only.

We guarantee that the stated settings are applicable for the materials provided directly by Kühling&Kühling or manufacturers explicitly named here. Other suppliers are available but since 3D printing filaments are far from being subject to manufacturing standards, Kühling&Kühling cannot be held liable for problems arising from filament obtained from non-specified sources.

Nonetheless we will provide assistance and support for any question concerning the handling/printing of materials. Please also refer to the [filament quality check procedure](#) and the respective sections of the [service guide](#) as well as the [troubleshooting guide](#) if you experience any trouble with your printing material. We are trying to provide information about and remedies for all known problems and are thankful for any hint on new methods, materials and solutions.

- [Acrylonitrile Butadiene Styrene \(ABS\)](#)
- [High-impact Polystyrene \(HIPS\)](#)
- [Polyamid \(PA, Nylon\)](#)
- [Polycarbonate \(PC\)](#)
- [Polyethylene Terephthalate \(PET\) Copolyester\)](#)
- [Polyvinyl Alcohol \(PVA\)](#)
- [Thermoplastic Elastomers \(TPE\)](#)
- [Polylactic Acid \(PLA\)](#)

Acrylonitrile Butadiene Styrene (ABS)

ABS is a thermoplastic polystyrene copolymerizate of amorphous micro-structure. It is very durable, rigid, tough and impact-resistant in a temperature range of -45°C to +70°C (+80°C for short-term exposure). The chemical resistance at normal ambient temperature is good against caustic solutions, organic acids, alcohols and fats.

ABS finds its uses in the automotive industry, the electrical industry, for toys (e.g the most famous LEGO® brick) and furniture. Covers of kitchenware and office machines as well as body panels in cars and seat shells are often made from ABS. It is the choice material for 3D printed functional parts of excellent dimensional stability, mechanical resilience, and usability as well as meaningful prototypes. ABS can be assembled by gluing and finished by grinding and sanding as well as painting and varnishing.

The material can be purchased as filament in a wide variety of colors including snow-white and black.

The HT500 has been built and optimized for printing ABS. The required settings are provided with the material profile *Kühling&Kühling ABS snow white* installed on the printer at delivery and with the config bundles for Slic3r available at the [GitHub repository](#).

Heating the build chamber is imperative when printing ABS since a uniform temperature distribution throughout the model is one key to minimize warping. If the entire model is kept at a temperature as near to the material's glass transition temperature (T_{eg}) as possible the cooling process can proceed much more evenly. To ensure firm sticking of the first layer, the print bed needs a temperature of at least 100°C.

Printing ABS on the HT500

Feature	Setting	see also
Print bed design	Standard PEI print bed	
Print bed temperature	100-110°C choose higher temperatures the bigger the parts are	
Build chamber temperature	70°C	
Extruder temperature	255-265°C	
Speed settings	As preset in the Slic3r profiles: - Perimeter: 25mm/s - Small perimeters: 10mm/s - External perimeters: 25mm/s - Infill/Solid infill/Top solid infill/Support material: 30mm/s - Bridges: 7mm/s - Gap fill: 300mm/s - Support material interface: 100% - Travel: 200mm/s - First layer speed: 15mm/s	GitHub
Idler lever preload	5.0-5.4mm	Setting the idler lever preload

Additional things to know

Feature	Setting	see also
Solvent	Acetone best efficiency when stirred; small amounts of ABS will be dissolved without mechanical interference, larger amounts will soften and can be scraped off or cut away; dissolve ABS remnants for blending ABS glue	Cleaning recommendation PC
Suitable soluble support material	HIPS will extrude at the same chamber/bed temperature settings and can be dissolved in d-limonene without altering the ABS properties	HIPS Dual extrusion
Warping	strong, accurately leveled print bed is mandatory	Warping prevention
Stringing	little to none	
Scaling factor to compensate for thermal expansion	approx. 101%	Thermal expansion

Feature	Setting	see also
Support requirements	Needs support structures for overhangs of more than 45°.	
Layer binding	strong	
Part strength/stability	strong, tough, medium flexibility	
Drying	not required	
Storage	in the open	

High Impact Polystyrene (HIPS)

HIPS is polybutadiene rubber-modified polystyrene, a semi-crystalline thermoplastic. It has good impact resistance and dimensional stability and is in its natural (translucent white) state FDA compliant.

HIPS is a low cost material that is easy to machine and fabricate, often used for low strength structural components and finds use in food processing applications, prototyping, toys, housings and covers. It is easy to fabricate, paint and glue.

Since HIPS sticks well to ABS, is thermally tolerant and can be dissolved in a medium that does not affect ABS, it makes for a perfect soluble support material for complicated, interlacing ABS objects. Both materials can be printed together in dual extrusion and with new and upcoming Slic3r developments (keep up to date on <http://slic3r.org/>), break-away support with all its disadvantages concerning surfaces will soon be a thing of the past.

Printing HIPS on the HT500

Feature	Setting	see also
Print bed design	Standard PEI print bed	
Print bed temperature	60-100°C ¹	
Build chamber temperature	20-70°C	Build chamber temperature
Extruder temperature	220-230°C	
Speed settings	use standard settings for ABS	ABS settings
Idler lever preload	5.0mm	Setting the idler lever preload

¹) When printing HIPS solely, the temperatures of print bed and build chamber can be lowered but HIPS withstands the higher temperatures needed for printing ABS which makes it an ideal soluble support material.

Additional things to know

Feature	Setting	see also
Solvent	d-limonene best efficiency when stirred USE RESPONSIBLY! <i>Limonene can cause skin irritations and is very toxic to aquatic organisms. Fumes are highly flammable. Wear skin protection when handling the solvent. Do not empty into drains. Do not use near open flames or heat sources.</i> <i>Observe the manufacturer's safety data sheet and make accessible to anyone handling Limonene!</i>	HIPS post-treatment
Warping	medium, good subsurface contact required	Warping prevention
Stringing	little to none	

Feature	Setting	see also
Support requirements	needs support structures for overhangs of more than 45°	
Layer binding	good to medium, depending on contact area	
Part strength/stability	medium, hard surface, thin structures are brittle	
Drying	not required	
Storage	in the open	

Polyamid (PA, Nylon)

Nylon used in 3D printing is a semi-crystalline thermoplastic of manifold technical uses. PA6 and PA6.6 have become famous in textile fabrication and gave „Nylon“ its worldwide reputation. They are both still very important in textile industries.

For 3D printing, PA6 and PA12 are most commonly used. They differ in their mechanical and chemical properties and applications.

PA6 has good dampening and wear characteristics, is thermostable up to +70 °C and impact resistant down to -40 °C. In combination with its high tensile strength it is used for various applications, such as bearings, coatings, housings, guitar strings and surgery suture material, motor parts and others. Its main disadvantage is its strong water absorption (up to 2.6 %) which induces swell and loss of tensile strength.

PA12 is the polyamid with the lowest water absorption (0.7 %) and therefore suitable for constructive parts in humid environment. It is not as strong and thermostable as PA6 (up to +50 °C) but chemically and impact resistant. PA12 is used as packing material in food and pharmaceutical applications, for cable insulation and gas and oil resistant hoses.

All polyamids are warping strongly and require adaptation of the print settings.

PA12

Printing PA12 on a HT500

PA12 produces strong and stable prints.

Feature	Setting	see also
Print bed	Standard PEI print bed	
Print bed temperature	130°C for the first layer (only!), then reduce to 100°C for the following layers.	Slic3r manual
Build chamber temperature	70°C	
Extruder temperature	285°C for the first layer (only!) then reduce to 270°C for the following layers.	Slic3r manual
Speed settings	Set first layer speed to 2mm/s; following layers can be extruded at standard ABS settings.	Slic3r manual ABS settings
Infill	25% rectilinear	
Solid layers Top/Bottom	3	
Perimeters	3	
Brim	2-5mm	
Support material contact Z-distance	0.15mm	
Idler lever preload	3.2mm or less	Setting the idler lever preload

Additional things to know

Feature	Setting	see also
Solvent	Formic acid 85% DANGER OF CAUSTIC BURNS! <i>Formic acid can cause severe burns at skin and eye contact. Fumes cause irritations of the respiratory tract.</i> <i>Wear adequate protective gloves and face/eye protection when handling formic acid.</i> <i>Always observe the manufacturer's safety data sheet and make it accessible to all persons handling formic acid!</i>	
Warping	strong	Warping prevention
Drying	recommended; dry at 60°C for 2-4 hrs.	
Storage	in closed lid container after drying, preferably with desiccant	

PA6

Printing PA6 on a HT500

Feature	Setting	see also
Print bed design	Tufnol with PVA glue	
Print bed temperature	110°C	
Build chamber temperature	70°C	
Extruder temperature	260 to 280°C	
Speed settings	Set first layer speed to 2mm/s; following layers can be extruded at standard ABS settings.	

Additional things to know

Feature	Setting	see also
Idler lever preload	3.2mm or less	
Solvent	Formic acid 85% DANGER OF CAUSTIC BURNS! <i>Formic acid can cause severe burns at skin and eye contact. Fumes cause irritations of the respiratory tract.</i> <i>Wear adequate protective gloves and face/eye protection when handling formic acid.</i> <i>Always observe the manufacturer's safety data sheet and make it accessible to all persons handling formic acid!</i>	
Warping	strong	Warping prevention
Drying	urgent; dry at 60-80°C for 2-8 hrs.	
Storage	in closed lid container after drying, preferably with desiccant	

Polycarbonate (PC)

PC is an amorphous thermoplastic of high strength, impact resistance, rigidity and hardness. Due to the micro-structure, PC is clear and colorless although colorized varieties are available. It finds its commercial use as a replacement for glass (airplane windows), in optical applications

(lenses), for housings and, its most famous application, CDs, DVDs, Blue-Rays. It is most commonly used where other plastics are too damageable.

PC has good insulation properties for electrical currents and is resistant against most mineral acids, oils and fats.

Extruding PC requires high temperatures since it has a high melting temperature and is warping strongly. The print bed must be pretreated with ABS glue.

3D printed PC is strong and stiff. Support material generated from PC is hard to remove due to the excellent layer binding characteristics and object design should regard this.

Printing PC with the HT500

Feature	Setting	see also
Print bed design	Standard PEI print bed	
Print bed temperature	First layer: 130°C Layers: 120°C	
Build chamber temperature	75°C	
Extruder temperature	First layer 285°C Layers: 275°C	
Speed settings	Reduce first layer speed to 5mm/s all other settings as standard presets for ABS	ABS settings
Idler lever preload	3.5mm	Setting the idler lever preload

*) Dissolve ABS remnants in Acetone until you get a thin, opaque liquid. Apply thinly by dappling the print bed with a soaked cloth.

Additional things to know

Feature	Setting	see also
Solvent	Acetone best results when stirred	
Warping	strong, needs sound contact to the accurately leveled print bed	Warping prevention
Stringing	medium	
Support requirements	Needs support structures for overhangs at 45°. Due to the strong layers, support is hard to remove without breaking delicate structures.	
Layer binding	very strong	
Part strength/stability	tough, hard and brittle	
Drying	urgent; dry at 80-90°C for 4-8 hrs.	
Storage	with desiccant in sealed lid container after drying	

Polyethylene terephthalate (PET) Copolyester

PET-Copolyester is a derivate of the thermoplastic PET. It is highly wear and abrasion resistant and chemically resistant to acidic solutions, chlorinated aqueous solutions (e.g. used to sanitize food processing equipment), fuels and fats.

PET-Copolyester is, like PET, commonly used for packagings, bottles and sheets in the food and medical industries as well as for displays and covers.

Printing PET-Copolyester with the HT500 is quite easy as the material is much less demanding than

ABS and can be printed at lower temperatures. It best sticks to the print bed at a temperature of 60°C and requires no build chamber heating at all, although heating does not negatively influence the print (which may be necessary for dual extrusion of different materials).

Warping is not an issue when printing PET-Copolyester. The printed surfaces are of high quality and the objects are dimensionally stable. Uncolored material stays transparent when 3D printed.

PET-Copolyester can be purchased as 3D printing filament under the brand name *XT-Coployester* directly from the manufacturer *ColorFabb* in a wide range of colors. It comes with multiple FDA certifications.

Printing PET-Copolyester with the HT500

Feature	Setting	see also
Print bed design	Standard PEI print bed	
Print bed temperature	60-80°C	
Build chamber temperature	20-50°C	Build chamber temperature
Extruder temperature	260-270°C ¹	
Speed settings	use standard presets for ABS	ABS settings
Idler lever preload	4.5mm	Setting the idler lever preload

¹ The higher the temperature, the more translucent clear material is extruded.

Additional things to know

Feature	Setting	see also
Solvent	Trichloroacetic acid DANGER TO HEALTH! <i>Trichloroacetic acid is highly caustic at skin contact and can cause respiratory tract irritation when inhaled. It is also very toxic to aquatic organisms.</i> <i>Always wear skin and eye protection when handling the agent. Observe the manufacturer's safety data sheet and make accessible to anyone handling the solvent.</i> <i>Do not empty into drains.</i>	
	Saturated steam DANGER OF BURNING! <i>Saturated steam has a temperature of at least +120 °C.</i> <i>Skin contact can cause severe burning injuries. Make sure that saturated steam cannot escape during the heating process.</i>	
Warping	strong, needs sound contact to the accurately leveled print bed	Warping prevention
Stringing	strong, cobweb-like; increasing with the extrusion temperature	
Support requirements	Can be extruded without support for overhangs up to 60°. Due to the strong layers, support is hard to remove without breaking delicate structures.	
Layer binding	strong	
Part strength/stability	tough, hard and flexible; clear material is brittle	
Drying	not required	
Storage	in the open	

Polyvinyl alcohol (PVA)

PVA is a water-soluble synthetic polymer that is used in papermaking, textiles, and a variety of coatings. As 3D printing filament, it has a yellowish clear appearance.

PVA has excellent film forming, emulsifying and adhesive properties. It is also resistant to oil, grease and solvents. It has high tensile strength and flexibility, as well as high oxygen and aroma barrier properties. However these properties are dependent on humidity, in other words, with higher humidity more water is absorbed.

PVA easily absorbs air moisture which leads to an increase in diameter that may cause jamming of the nozzle and affects prints negatively by degassing.

Absorption of water reduces the tensile strength, but increases elongation and tear strength.

INFO



We recommend drying PVA for 2-3 hours at 60°C before printing the first time and store it in a sealed container with drying granulate.

3D printed PVA objects are stiff and brittle, the layer binding is excellent and the warpage minimal. PVA is a useful support material for [PETG](#) since both materials can be extruded at equal temperature settings and stick viably together.

Printing PVA with the HT500

Feature	Setting	see also
Print bed design	Standard PEI print bed	
Print bed temperature	max. 60°C	
Build chamber temperature	30-40°C	Build chamber temperature
Extruder temperature	205-215°C	
Speed settings	use standard presets for ABS	ABS settings
Idler lever preload	3.5mm	Setting the idler lever preload

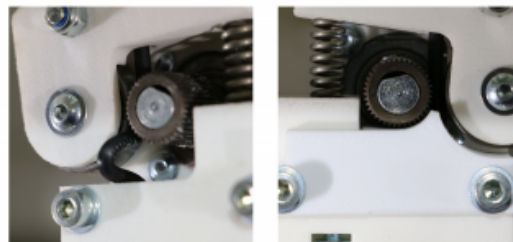
Additional things to know

Feature	Setting	see also
Solvent	Water best solving properties when heated to 30 to 40°C and stirred.	
Drying	required; dry at 60°C for 3-4 hrs.	
Storage	in closed lid container after drying, preferably with desiccant	

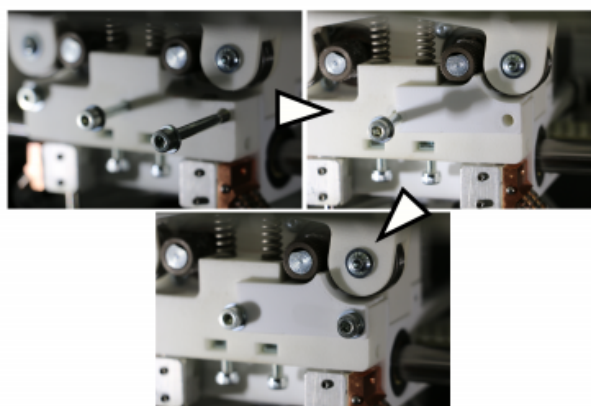
Thermoplastic elastomer (TPE)

TPEs are a group of elastomers suited to be printed with 3D printers. They come in different mixtures, according to the basic substances used. There are polyolefin blends (TPE-o commonly simply called TPE), polyurethane-based TPE-u (or TPU), styrenic block copolymers (TPE-s) and more. They all provide elastic properties and can be used for printing dampers, seals or plugs and a lot of other parts that need elasticity and flexibility.

Printing TPEs cannot be done without some alterations of the HT500's standard setup. First thing needed is an adapter (provided for download at [Kühling&Kühling GitHub](#)) to bridge the gap between the extruder drive gear and the hot-end inlet otherwise the flexible material will twist and bent. Also, the standard print bed will not suffice and some of the Slic3r settings must be adapted.



 Flexible filament bending and twisting without and extruding smoothly with the adapter installed.



 Installing the adapter for flexible filaments.

Printing TPE with the HT500

Feature	Setting	see also
Print bed design	TPE-o: Epoxy-resin based board (e.g. FR4)	
	TPE-u: standard PEI print bed	
Print bed temperature	TPE-o: 105 °C	
	TPE-u: 70 °C	
Build chamber temperature	max. 40°C	Build chamber temperature
Extruder temperature	TPE-o: 230-240°C	
	TPE-u: 225-240°C	
Speed settings	- massively reduce print speed: max. 7 m/s for every print operation (all perimeters, infill, support etc.) or volumetric speed 1 mm ³ /s (available since Slic3r v1.2.8) - reduced priming speed	Reducing the priming speed
Idler lever preload	4.3mm	Setting the idler lever preload
Nozzle tip	0.5 or above	

Additional things to know

Feature	Setting	see also
Solvent	None Elastomers do not stick to metal or other plastic surfaces easily and can simply be pulled off for most cleaning purposes.	
Warping	low	
Drying	not required	
Storage	in the open	

Polylactic Acid (PLA)

PLA, a polyester derivate that is due to its molecular structure biodegradable in composting plants, is one of the most common 3D printing material since it is easy-to-use and very rewarding. It is mostly used in 3D printing for visualization of organic forms, such as architectural and landscape models, busts, and decoration. Industrial applications of PLA are found in farming (mulch foils), medical engineering (implants and fixation, seam material), food packaging (yogurt pots), and the textile industry; its UV-resistance, high flashpoint and colorfastness make it useful in a variety of other applications.

The mechanical properties of PLA are near those of Polystyrene (PS) but with a lower glass transition and continuous operation temperature (45 - 65 °C), which makes it unusable for exposition to heat (e.g. coffee mugs).

PLA is available from a lot of manufacturers in a variety of colors and different fillers (e.g. wood or bronze fill), which makes it easily available, relatively cheap and suitable for many visualization applications. Due to the high availability it is not part of the Kühling&Kühling product range.

Printing PLA with the HT500

Feature	Setting	see also
Print bed design	standard PEI print bed, glass, BuildTak	
Print bed temperature	OFF-50°C	
Build chamber temperature	OFF°C	Build chamber temperature
Extruder temperature	180-220°C *	
Speed settings	reduce standard settings for ABS by 20% * Perimeters: 20mm/s Infill: 24mm/s	
Idler lever preload	4.0mm	Setting the idler lever preload
Nozzle tip	all	

* depending on color, filler; test to find best suitable settings for your application.

For example: [colorFabb's](#) famous *woodfill* is best extruded at 225 °C with 12 mm/s for perimeters and 24 mm/s for the infill.

Additional things to know

Feature	Setting	see also
Solvent	Tetrahydrofuran USE RESPONSIBLY! <i>Tetrahydrofuran can cause eye irritations and irritations of the respiratory system. Fumes are easily flammable.</i> <i>Apply skin protection cream or wear protective gloves when handling the solvent. Do not empty into drains. Do not use near open flames or heat sources.</i> <i>Observe the manufacturer's safety data sheet and make accessible to anyone handling tetrahydrofuran!</i>	
Layer binding	good	
Part strength/stability	medium; parts are stiff and brittle	
Warping	low	
Drying	not required	
Storage	in the open	

High-temperature materials

For extruding the materials described in the following temperatures above 300°C are required. For 3D printing them on a HT500 this means that the (optional) high-temperature hot-end is required, which allows for extrusion temperatures of up to 400°C.

- [Polyether ether ketone \(PEEK\)](#)

Polyether ether ketone (PEEK)

PEEK is a semi-crystalline high performance thermoplast resembling Polyoxymethylene (POM) or PET but with better mechanical properties and temperature stability. Also, it features a high solidity, stiffness, chemical and radiation resistance, high flaming point and good sliding and wear properties. Due to its superior properties, the material is very expensive, which often makes it second choice behind POM or similar.

PEEK is typically used for bearings, thrust washers, gear wheels, gaskets and pumps.

Printing PEEK on a HT500

Feature	Setting	see also
Print bed design	Standard PEI print bed	
Print bed temperature	130°C	
Build chamber temperature	70°C	
Extruder temperature	370-395°C	
Speed settings	Reduced speeds; approx. 70% of the standard settings for ABS.	
Idler lever preload	1mm	Setting the idler lever preload

Additional information

Feature	Setting	see also
Warping	low	
Stringing	little to none	
Drying	not required	
Storage	in the open	
Tempering	can be useful for very stiff material that keeps uncoiling; heat at 200°C for 1-2 hrs.	
FDA and EMA compliant		

Tips & Tricks

Here you will find information about issues concerning the operation of the HT500 in your specific work environment or solutions for questions that have been upraised by support requests.

- [Command line access](#)
- [Setting a static IP address](#)
- [Setting time signals](#)
- [Password change](#)
- [Leveling tips](#)
- [Switching heating elements off via the G-code](#)
- [Changing the build chamber temperature via the G-code](#)
- [G-code commands for direct use at the 3D-printer](#)

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 HT500s' 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 HT500 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
```

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
broadcast 192.168.1.255
gateway 192.168.1.1
dns-search family.local
dns-nameservers 192.168.1.1
pre-up iptables-restore </etc/iptables.rules
```

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](#) and [reboot](#) the HT500 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 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](#) and [reboot](#) the HT500 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).

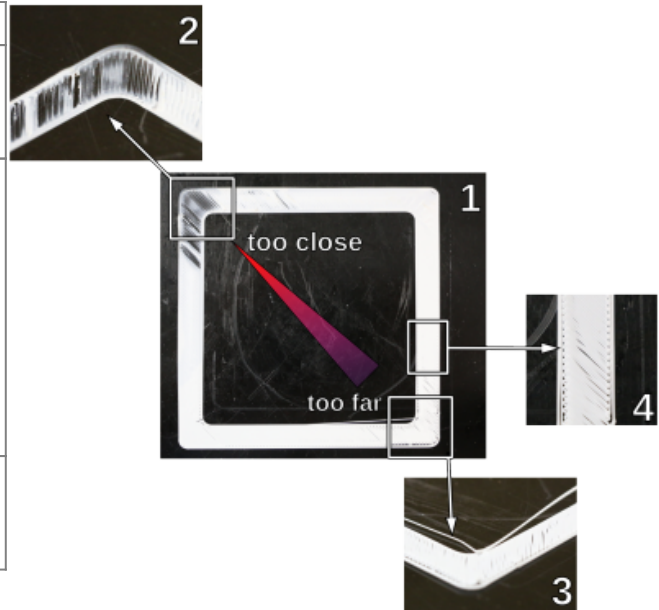
Print bed leveling

Accurate leveling is vitally important for the print result. Although correct first layer settings can compensate for slight unevennesses of the print bed, false leveling will ruin a print within the first few layers.

Evidence for a leveling mistake can be:

No.	Appearance	Reason
1	Asymmetrical layer thickness, especially of the first layer.	lopsided leveling Appearance 2 and 3 are visible simultaneously

No.	Appearance	Reason
2	Smearing of the extrusion and possibly clogging of the nozzle.	Print bed and the nozzle are too close together.
3	The extruded strand is laid on the print bed instead of being spread. Strands do not stick to the print bed but are being pulled away by the nozzle tip.	Print bed and the nozzle are too far apart. (can be due to a wrong bed temperature also - better double-check)
4	Extrusion of rounded, unjoined strands.	Print bed and the nozzle are too far apart.



Leveling mistakes: examples of characteristic appearances of the first layer.

To avoid such irritations, make use of the following tips:

[Leveling test is OK](#) [Leveling test is not OK](#)

- Always make sure that the build chamber is adequately and uniformly preheated to at least 90 % of the target temperature (e.g. 63 °C for ABS).
The Z-end stop is temperature sensitive and may cause deviating positioning when exposed to temperature variations.
Keep the build chamber doors as shortly open as possible during leveling to avoid too much heat loss.
- If you removed the print bed, always re-insert it in the same direction and orientation it had during leveling.
- Put the print bed with the convex side down onto the print table.
- After adjusting all three leveling points, return the print head to the first position (tap the **[Back]** button). Check the bed - tip distance by tapping (**not** pressing) the print bed with a finger next to the nozzle tip: if you see a gap appearing at the slightest touch, the leveling point is correctly adjusted. If even the least pressure is required, the leveling is too high. Repeat this at the other two leveling points. Have a look at the adjacent videos for visual explanation.
- Before waisting time on unsuccessful

leveling, please rethink if the currently installed nozzle tip is adequate for the print job. Wider nozzles are more tolerant when it comes to compensating leveling mistakes and may be more suited to the task, especially when printing larger parts.

Tips for easier leveling

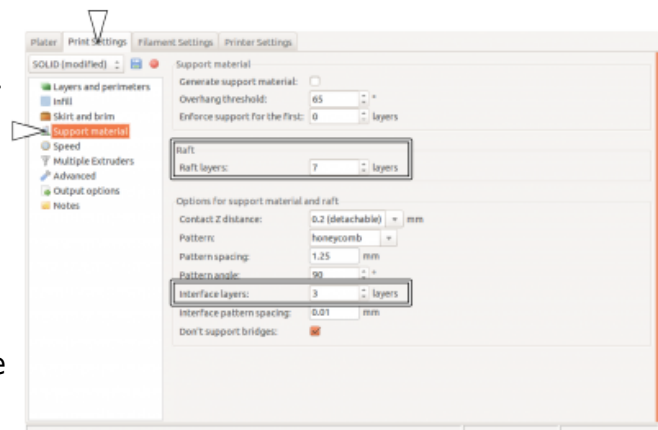
If you find it difficult to adjust the three leveling points uniformly with only the spring pressure, place a sheet of paper (not more than standard 80 g/m²) between the nozzle tip and the print bed and carefully push the bed against the tip before fastening the set screws. This way, you ensure a uniform bed - tip distance at all three leveling points.



Manual leveling with paper spacer - front and hind right leveling point.

Another way of making leveling a little bit less important is „floating“ your print on a raft. Slic3r (and most other slicing software) provides the „raft“ function, a special kind of support material, which means that before starting with the actual object a (customizable) number of loosely printed layers is printed to compensate slight leveling mistakes. If you choose the raft settings correctly, beginning with a layer as wide and thick as allowable (depending on the nozzle tip's diameter) leveling will become a much less annoying process.

The raft's first layer is treated as the print's first layer and uses the respective settings. The



Adding a freely chosen number of raft layers to the print.

Deactivating the heating elements after end of print job

1. Start your slicing software.
2. Open the [Printer Settings] tab.
3. Choose the „Custom G-code“ menu.
4. In the *End G-Code* editor scroll down to the end of the text field and position the cursor in the last line above the ; /END-GCODE entry.
5. Enter the command
`M104 S0 T2`
This will set the heating elements of the build chamber to a temperature of 0 °C.
6. Enter the command
`M140 S0`
This will deactivate the print bed as the last action of the current G-code.



Switching off the build chamber heating 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 HT500 such as stepper motors, bearings and electronics.

Since hitherto the HT500 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 HT500 is set via the „Start G-code“ which can be manually altered.

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

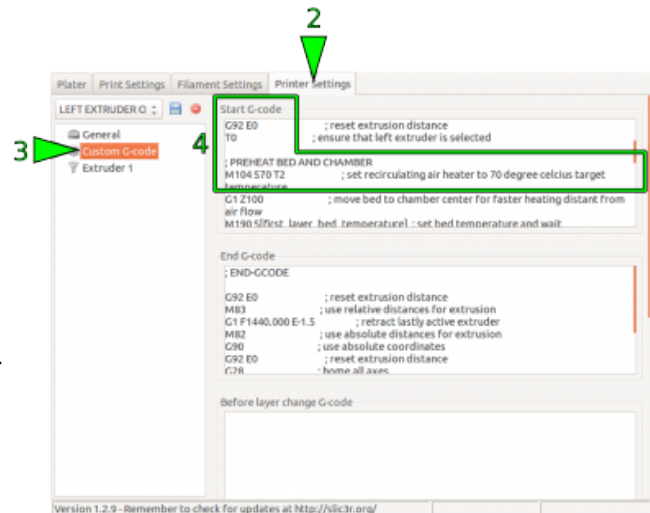
1. Open your slicing software.
2. Open the tab [Printer Settings].
3. Choose the *Start G-code* editor in the „Custom G-code“ menu.
4. Scroll down to
; PREHEAT BED AND CHAMBER
and position the cursor in the line reading:

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

5. 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
```

(for logical reasons, the comment should be aligned).



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

- If you want to keep the settings, save them in a new 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.




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>	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>	Set temperature without wait	Adjusting the build chamber temperature, Deactivating the heating elements after end of print job
M109 S<temp> T<extruder>	Set temperature with wait	
M140 S<temp>	Set bed target temp without wait	
M190 S<temp>	Set bed target temp with wait	
M221 S<extrusion flow multiplier in percent>	Increase/decrease given flow rate	M221 S95 → decrease flow to 95 % of g-code value
M220 S<print speed multiplier in percent>	Increase/decrease print speed of all drive speeds	M220 S95 → decrease print speed to 95 % of g-code value