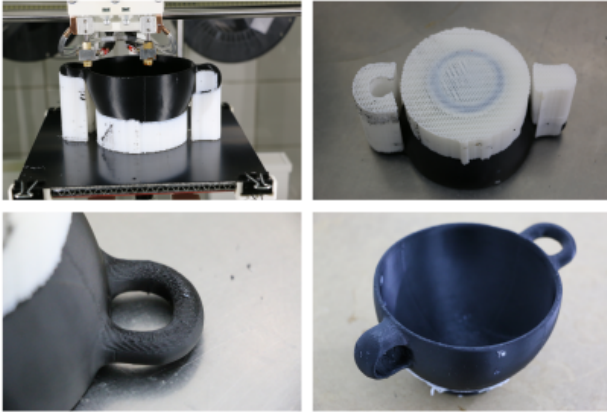


Dual extrusion



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

One of the RepRap Industrial 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.



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

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.*

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.

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.

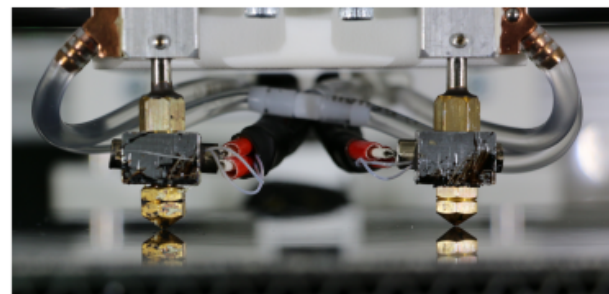
Wiper check

As previously mentioned, the RepRap Industrial 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 instructions provided at delivery. If otherwise, find the [instructions](#) here.



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.



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

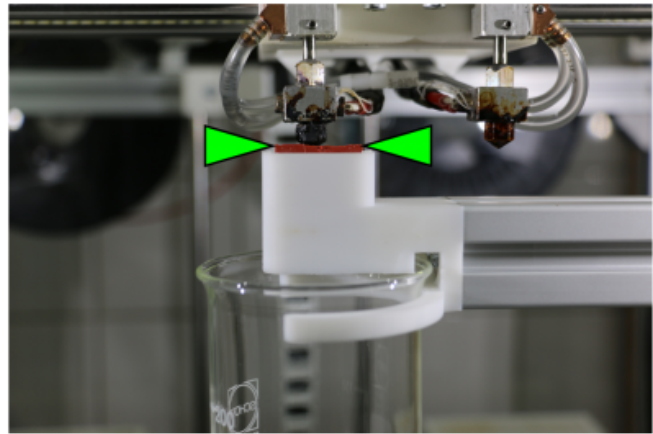
Check that the wiper lip and the nozzle tips are aligned correctly:

- 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.



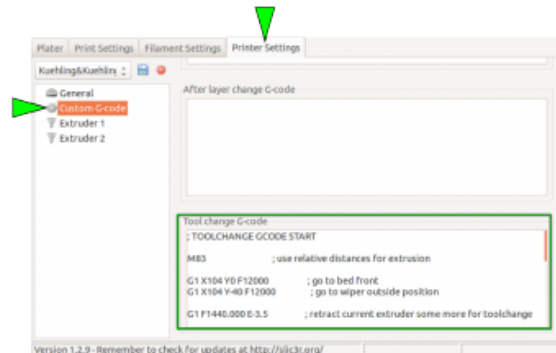
INFO

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 *Plater* 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](#).

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.

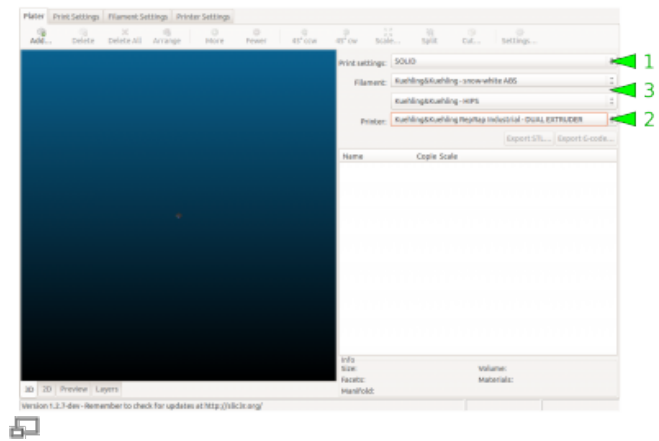
3. Filament

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"

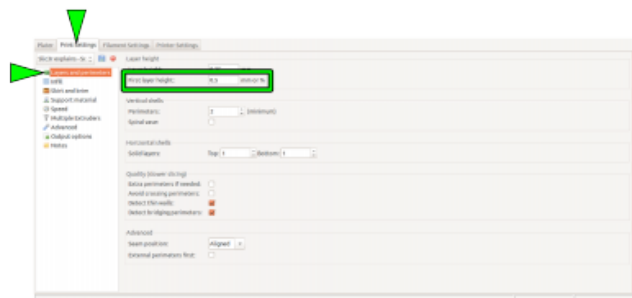
→ Layers and perimeters

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

INFO



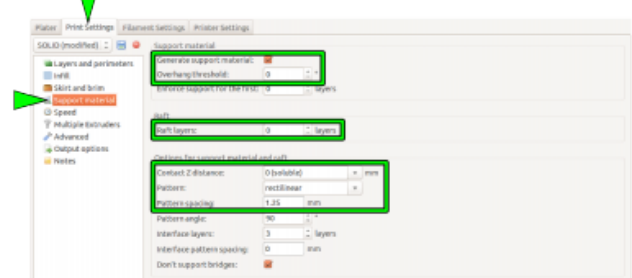
This function cannot be used with any ECO profile or similar settings because the solvent for the support material may soak into the model which can cause long-term damage.



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

→ Support material

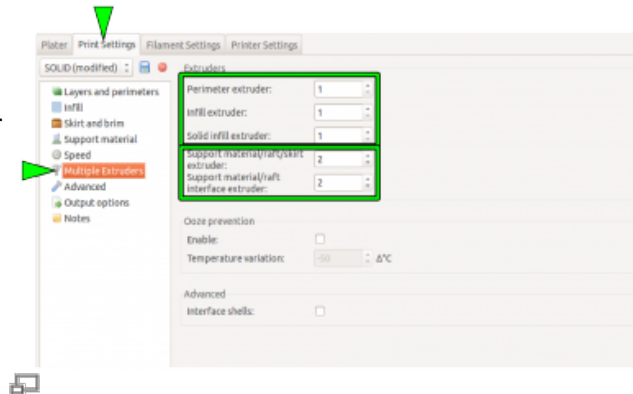
- The checkbox Generate support material must be activated. If required, the Overhang threshold angle can be adjusted accordingly (check via the Preview in the Plater).
- The Contact Z distance must be "0 (soluble)" with a (recommended) "rectilinear" Pattern and a Pattern spacing of 1.25 mm.
- It is 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.

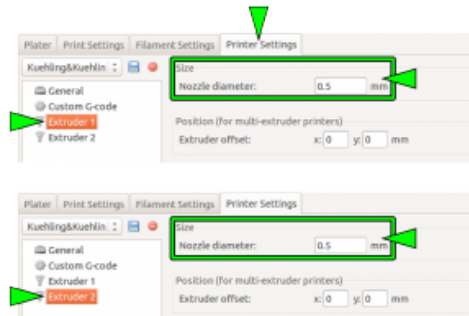


Assigning the hot ends to their respective tasks.

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!

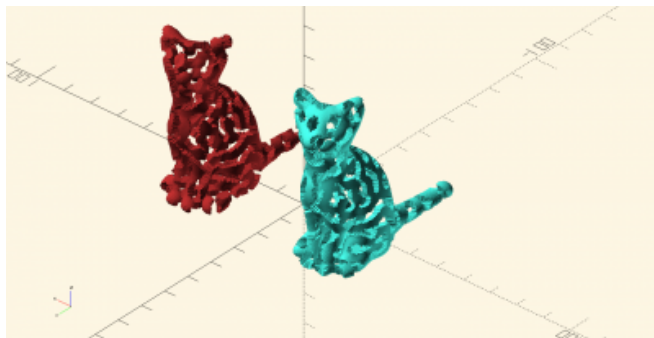
d-limonene is a solvating agent and therefore must not be handled carelessly. d-limonene can cause skin and eye irritations. Wear suitable protective gloves and goggles and regard the manufacturer's safety data sheet

when handling d-limonene.

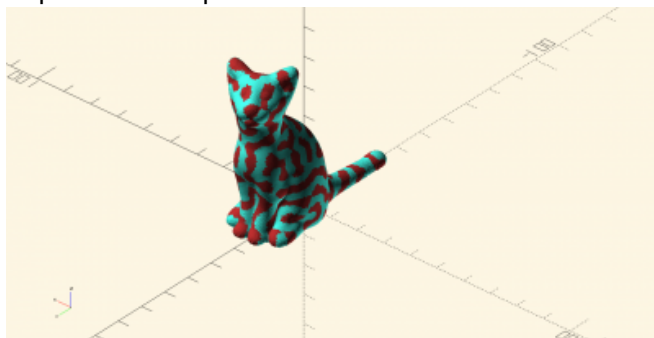
After a complex ABS model has successfully been printed with HIPS soluble support material, the support material must be removed and dissolved. This is an easy process that only requires only a few steps:

- Mechanically break away the easily accessible parts of the support material.
- Place the model in a bath of [Kühling&Kühling wash-out solution](http://www.kuehlingkuehling.de) (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 solving 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.

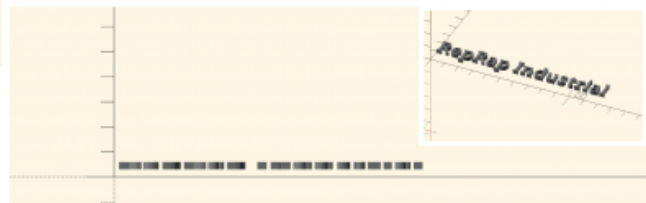
Bicolored printing



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.



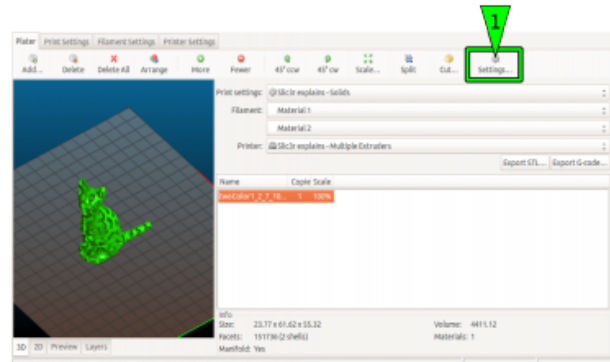
3D model preparation for bicolored printing
example 2: both individual parts of the 3D model
being aligned in a common assembly and
exported as separate STL-files with the correct
relative coordinates.

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.

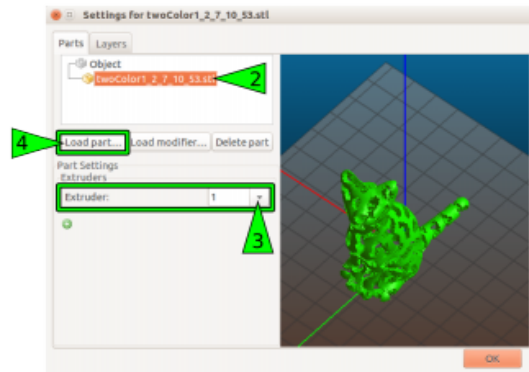
Uploading and slicing the parts:

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



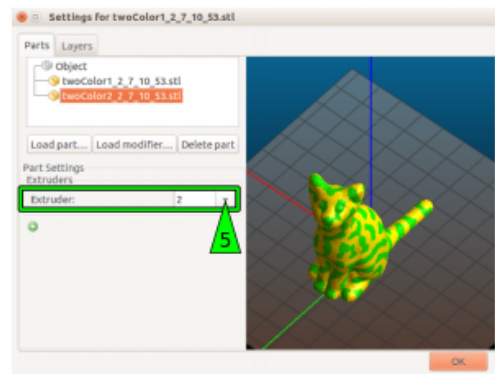
The first part is loaded to the Plater directly.

- 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.



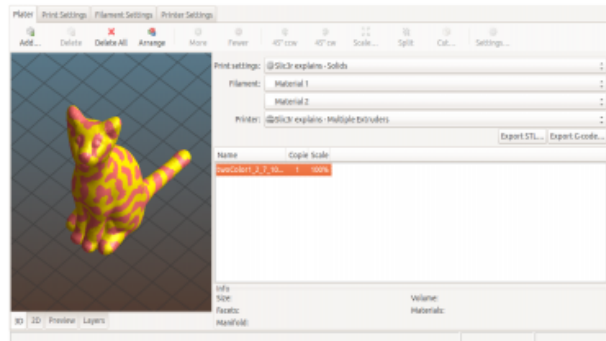
After the extruder for the first part has been assigned, the second part is loaded via the *Settings* dialog.

- 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.

Materials

All information given in the following have been thoroughly tested on the RepRap Industrial 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.

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 RepRap Industrial 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.
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



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.



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.

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 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.



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.

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.

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

$2.85 \pm 0.1 \text{ mm}$ (2.75 ... 2.95 mm)

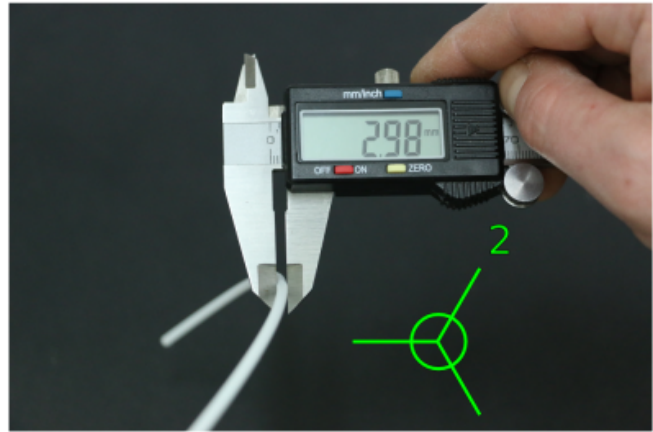
A diameter of e.g. 2.98 mm will increase friction in the filament feed system and the nozzle barrel and cause compression and jamming of the filament.

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

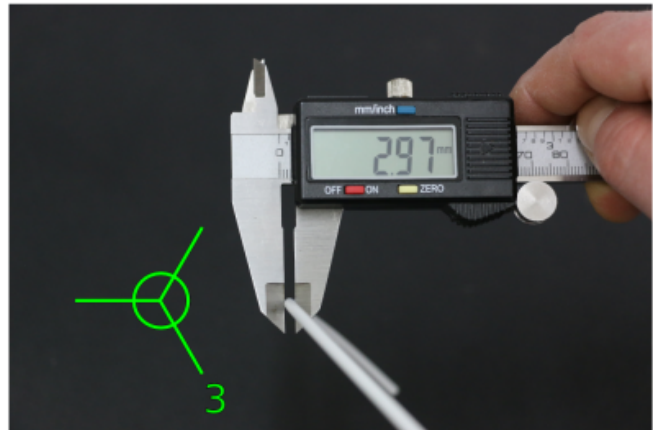
Filament below 2.75 mm is, especially when a print job requires a lot of retraction movement, likely to jam the nozzle barrel because molten filament can push itself back up the barrel and form a plug.



First measured diameter.



Second measured diameter.



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

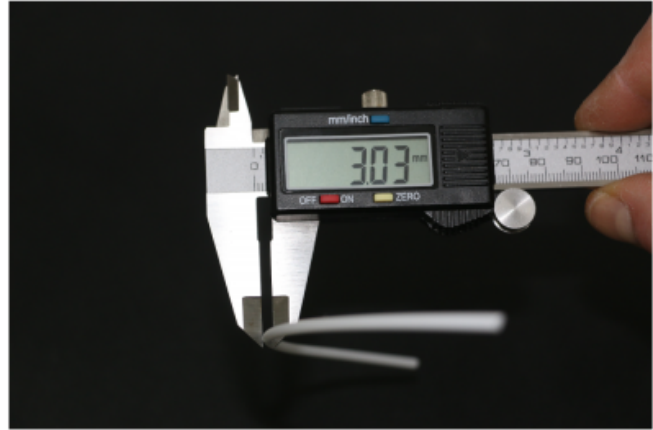
Roundness

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.



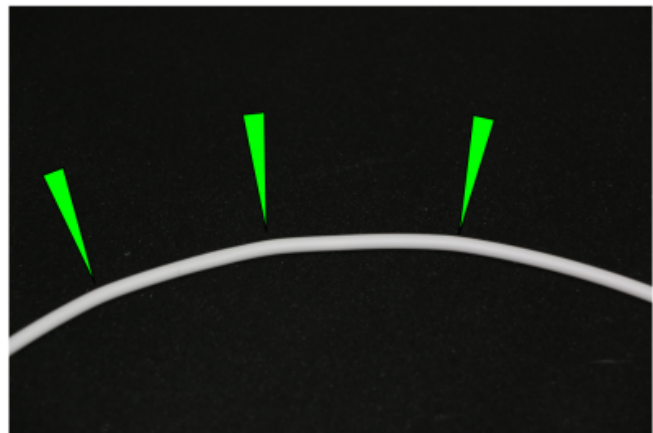
The measured diameter is below the admissible tolerance ...



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

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.



Visible kinks in the filament may also lead to jamming.

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 RepRap Industrial 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.

Print ABS on the RepRap Industrial with the following settings

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 (GitHub).
Idler lever preload	5.0 - 5.4 mm

Additional things to know

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; see service guide for cleaning recommendation of print bed, nozzles and barrels; dissolve ABS remnants for blending ABS glue (see 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. Find more information in HIPS and Dual extrusion
Warping	strong, accurately leveled print bed is mandatory	
Stringing	little to none	
Scaling factor to compensate for thermal expansion	approx. 101 %	
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.

Print HIPS on the RepRap Industrial with the following settings

Print bed design	Standard PEI print bed
Print bed temperature	60 - 100 °C ¹
Build chamber temperature	20 - 70 °C ²
Extruder temperature	220 - 230 °C
Speed settings	Use settings for ABS preset in the standard Slic3r profiles.

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.

²) To adjust the build chamber temperature follow the description provided [here](#).

Additional things to know

Idler lever preload	5.0 mm		
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. Observe the manufacturer's safety data sheet and make accessible to anyone handling Limonene!</i>
Warping	medium, good subsurface contact required		
Stringing	little to none		
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. PA 6 and PA 6.6 have become famous in textile fabrication and gave "Nylon" its worldwide reputation. They are both still very important in textile industries.

For 3D printing, PA 6 and PA 12 are most commonly used. They differ in their mechanical and chemical properties and applications.

PA 6 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.

PA 12 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 PA 6 (up to +50 °C) but chemically resistant and impact resistant. PA 12 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 need a special print bed or treatment of the print bed to get them sticking.

PA 6

Temperatures for PA 6 extrusion should be set as follows:

Print bed design	Standard PEI print bed or phenolic-paper circuit board, either thickly coated with well-dried ¹ PVA glue stick (e.g. 2 to 3 layers of UHU stic)
Print bed temperature	20 - 30 °C
Build chamber temperature	OFF ²
Extruder temperature	245 to 265 °C
Speed settings	Use speed settings for ABS preset in the standard Slic3r profiles.

INFO



We do not recommend printing PA 6 since we find that layer binding and overall part strength tends to differ strongly between various PA 6 filament suppliers and no universal statement about PA 6 advantages can be made in regard to FFF 3D printing at the moment. Still, the very best results have been achieved with Taulman's Nylon 618.

Additional things to know

Idler lever preload	3.2 mm or less	
Solvent	???	
Drying	required; dry at 60 - 80 °C for 2 - 8 hrs.	
Storage	in closed lid container after drying, preferably with desiccant	

PA 12

PA 12 produces strong and stable prints. Temperatures for PA 12 extrusion should be set as follows:

Print bed design	Tufnol with PVA glue
Print bed temperature	OFF
Build chamber temperature	OFF ²
Extruder temperature	240 to 250 °C

Speed settings	Use speed settings for ABS as preset in the standard Slic3r profiles.
----------------	---

Additional things to know

Idler lever preload	3.2 mm or less	
Solvent	???	
Drying	required; dry at 60 - 80 °C for 2 - 8 hrs.	
Storage	in closed lid container after drying, preferably with desiccant	

¹ Dry for at least 3 hours at room temperature. Applying heat (~100 °C) reduces drying time to 15 - 20 minutes.

² To adjust the build chamber temperature follow the description provided [here](#).

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.

Choose the following settings for PC extrusion with the RepRap Industrial

Print bed design	Standard PEI print bed with ABS glue*
Print bed temperature	120 °C
Build chamber temperature	75 - 80 °C
Extruder temperature	275 - 280 °C
Speed settings	Use presets for ABS in the standard Slic3r profiles.

*) 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

Idler lever preload	3.5 mm	
Solvent	Acetone	Best results when stirred.
Warping	very strong, needs sound contact to the accurately leveled print bed	
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 110 - 120 °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 RepRap Industrial 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-Copolyester* directly from the manufacturer *ColorFabb* in a wide range of colors. It comes with multiple FDA certifications.

Use the following settings for PET-Copolyester extrusion:

Print bed design	Standard PEI print bed
Print bed temperature	60 - 80 °C
Build chamber temperature	20 - 50 °C
Extruder temperature	260 to 270 °C ¹
Speed settings	Use presets for ABS in the standard Slic3r profiles.

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

Additional things to know:

Idler lever preload	4.5 mm	
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. 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. Do not empty into drains.</i>
	Saturated steam	DANGER OF BURNING! <i>Saturated steam has a temperature of at least +120 °C. 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	
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.

Choose the following settings for extruding PVA with the RepRap Industrial

Print bed design	Standard PEI print bed
Print bed temperature	max. 60 °C
Build chamber temperature	30 - 40 °C
Extruder temperature	205 - 215 °C
Speed settings	Use presets for ABS in the standard Slic3r profiles.

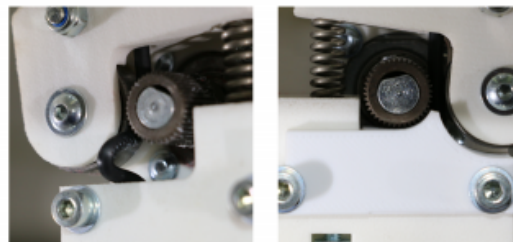
Additional things to know

Idler lever preload	3.5 mm		
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		

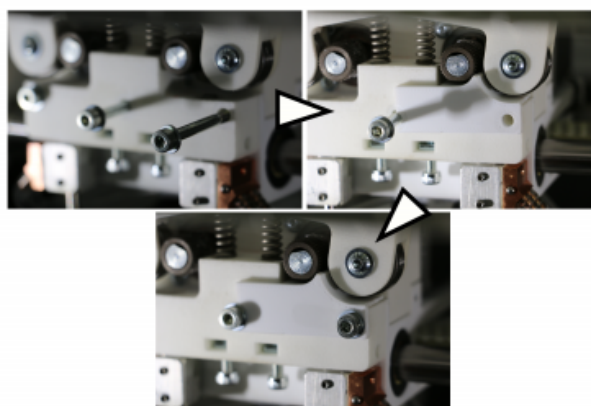
Thermoplastic elastomer (TPE)

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 RepRap Industrial'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.

Choose the following settings for extruding TPE with the RepRap Industrial

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
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 (see here)

Additional things to know

Idler lever preload	4.3 mm
Nozzle tip	0.5 or above
Solvent	None Elastomers do not stick to metal or other plastic surfaces easily and can simply be pulled off for most cleaning purposes.
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.

Choose the following settings for extruding PLA with the RepRap Industrial

Print bed design	standard PEI print bed, glass
Print bed temperature	40 - 60 °C
Build chamber temperature	20 - 40 °C
Extruder temperature	180 - 235 °C *
Speed settings	- reduce print speed (max. 20 mm/s for all perimeters) *

* 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

Idler lever preload	4.0 mm	
Nozzle tip	0.5 or above	
Solvent	Tetrahydrofuran	USE RESPONSIBLY! <i>Tetrahydrofuran can cause eye irritations and irritations of the respiratory system. Fumes are easily flammable. 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. Observe the manufacturer's safety data sheet and make accessible to anyone handling tetrahydrofuran!</i>
Layer binding	good	
Part strength/stability	medium	
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 RepRap Industrial 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)

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 RepRap Industrial:

Print bed design	Standard PEI print bed
Print bed temperature	130 - 140°C
Build chamber temperature	75°C
Extruder temperature	370 - 395°C
Speed settings	Reduced speeds; approx. 70% of the standard settings for ABS.
Idler lever preload	1mm

Additional information:

Warping	low
Stringing	little to none
Drying	not required
storage	in the open
FDA and EMA compliant	

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 RepRap Industrial.

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 RepRap Industrial, 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 RepRap Industrial 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 RepRap Industrial 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 RepRap Industrial 3D printer can also be found in the [Materials'](#) descriptions.

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

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, Kühling&Kühling 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 RepRap Industrial. We have taken all this into respect when creating our slicing profiles and you do not need to try and find matching



INFO

The following applies to the nozzle tip size 0.35 mm and the preset

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 RepRap Industrial we recommend the following (and have already taken it into respect in our slicing profiles):



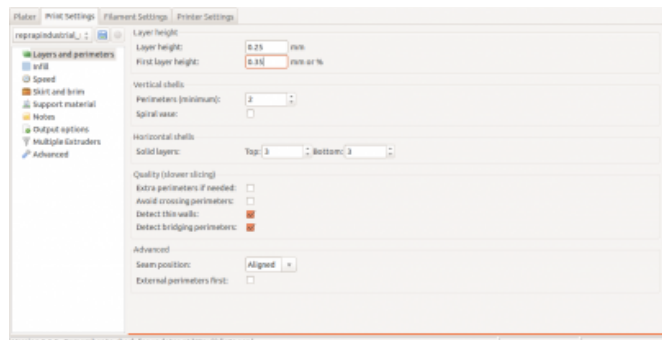
Slic3r profiles you installed during initial commissioning. It is meant to give you an overview of the settings relevant for the optimization of the first layer adhesion. Please experiment to find the settings best suited for your task and test all settings before applying them to your final model if 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.

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

For example:

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

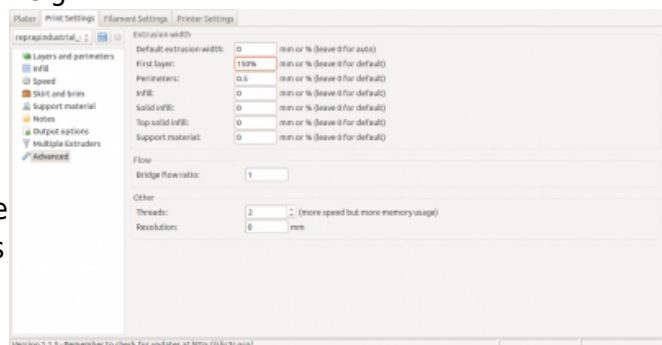


Slic3r Print Settings - setting the first layer height

- 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.



Slic3r Advanced Settings - reducing the first layer extrusion width

INFO

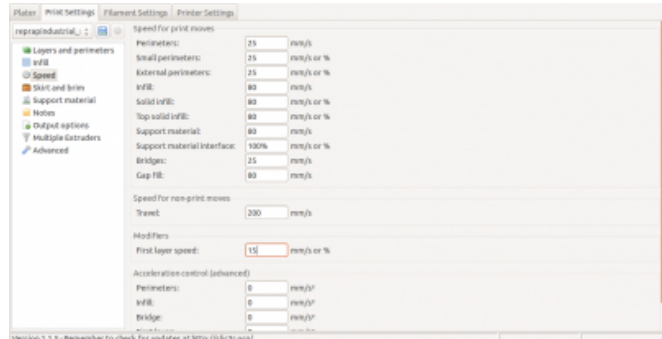


Both above mentioned measures compensate for unevennesses of the print bed and increase the tolerance against slight leveling mistakes. The distance between nozzle tip and print bed will assure that the bore is not clogged when



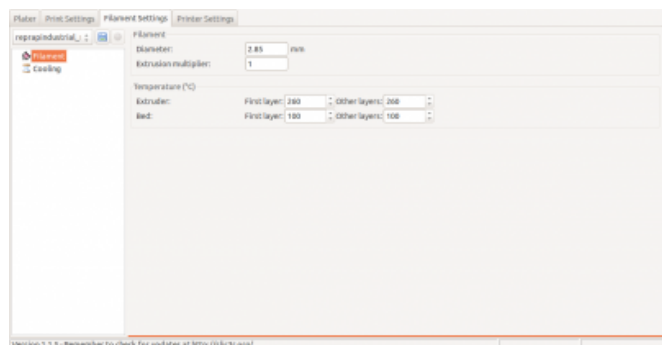
passing convex bumps by leaving enough free space in between. The extrusion width will make sure that enough material is conveyed to equalize concave crudities.

- 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).



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 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.



Slic3r Filament Settings - setting the first layer extrusion temperature

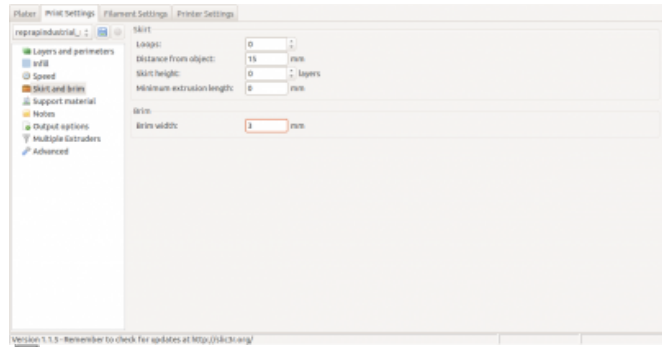
INFO



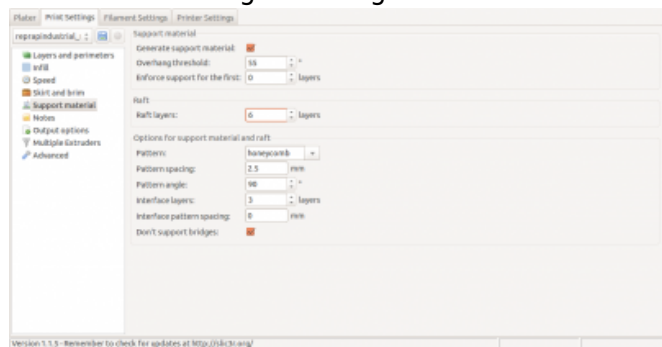
Optimize the extrusion and print bed temperature for the **Material** currently installed. Do not hesitate to try different settings here, since even colored additives may alter the optimal extrusion temperature.

- 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](#).



Slic3r Print Settings - adding a brim



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 RepRap Industrial 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 RepRap Industrial 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 RepRap Industrial 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.

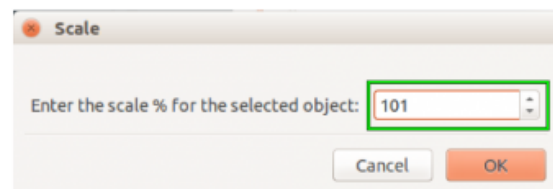
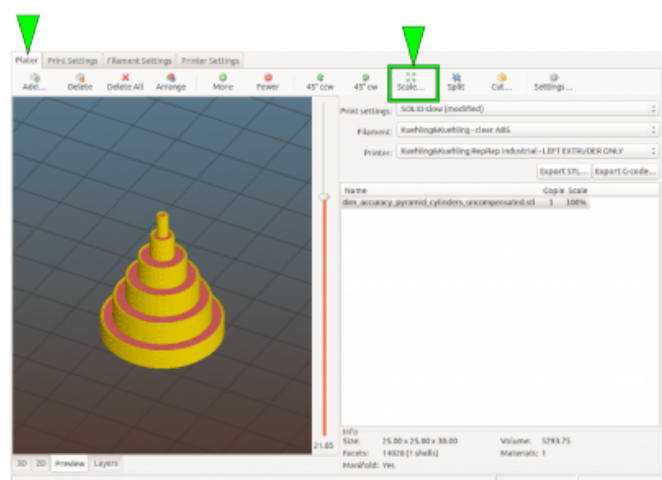
This method is advantageous if you want to check dimensions in the CAD-file, which is much more common for .stl-data than ready-sliced g-codes.

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_L)	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

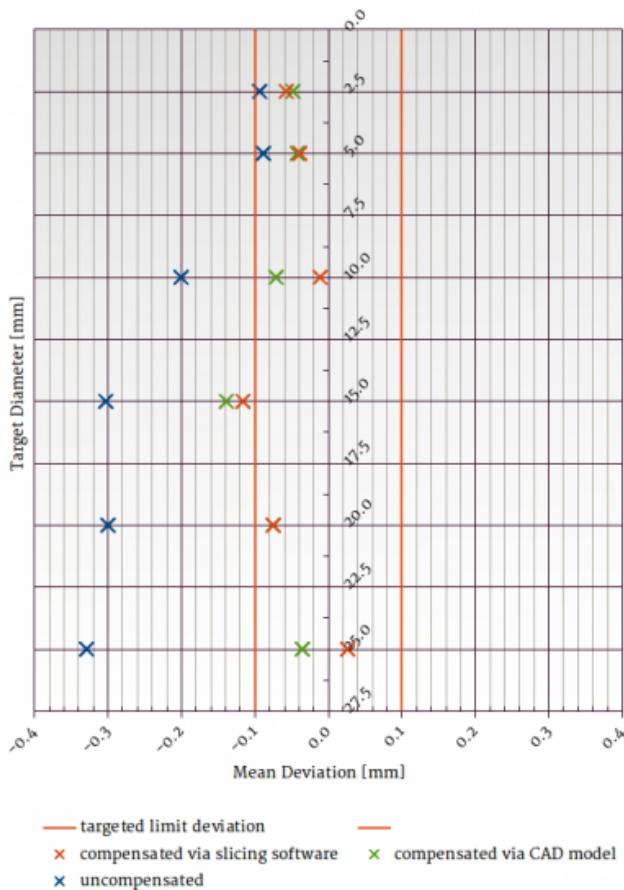
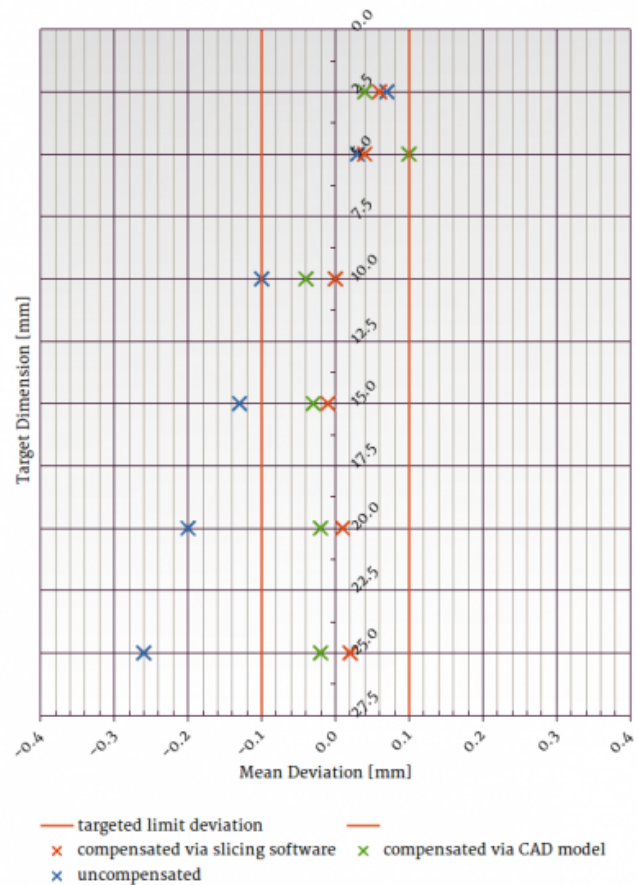


Illustration for Linear Dimensions

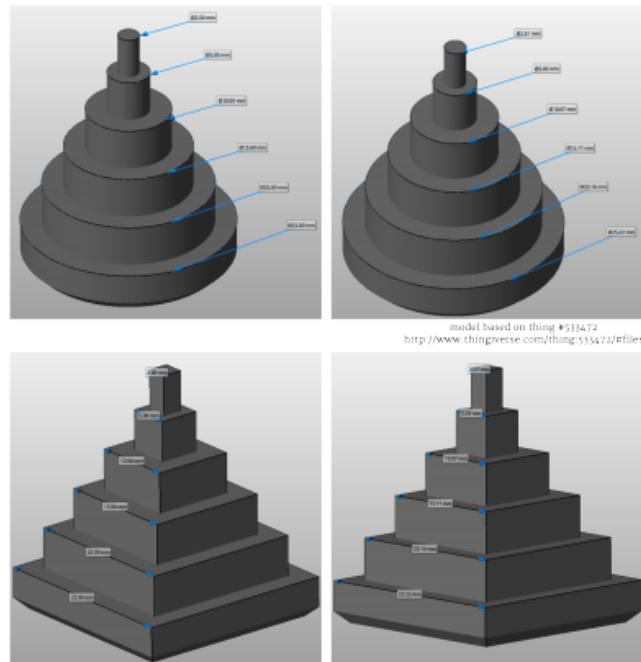


The deviation decreases significantly when the thermal expansion is compensated. Either for diameters of circular structures ...



... 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.

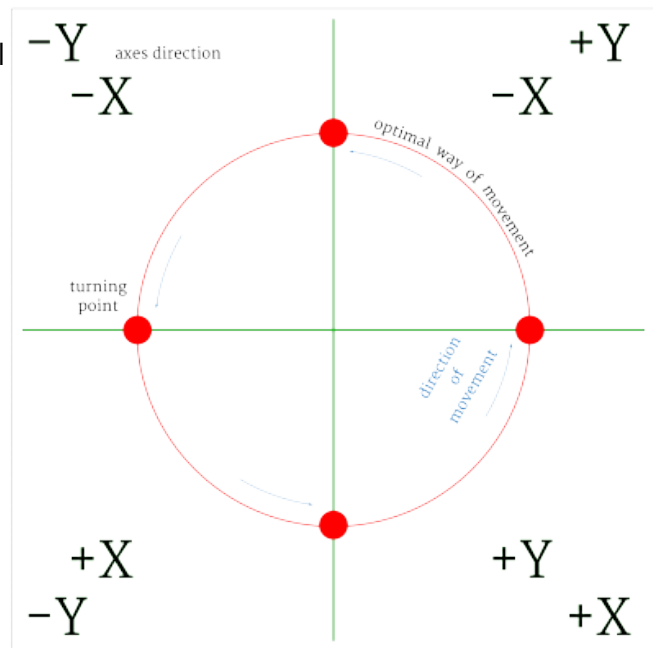


netfabb visualization of the ABS test bodies printed for comparison and evaluation. On the left hand-side the uncompensated model, on the right-hand side the model after multiplying with

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.



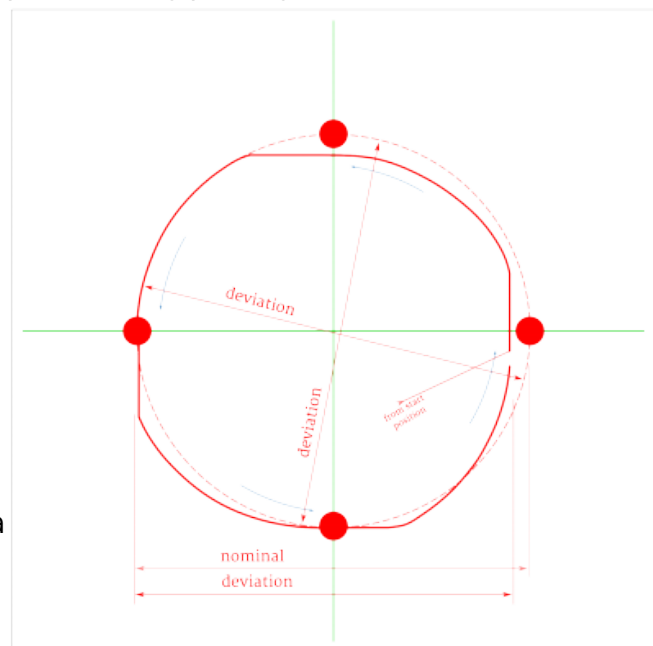
The four directions of movement and the turning points during printing a circle.

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.

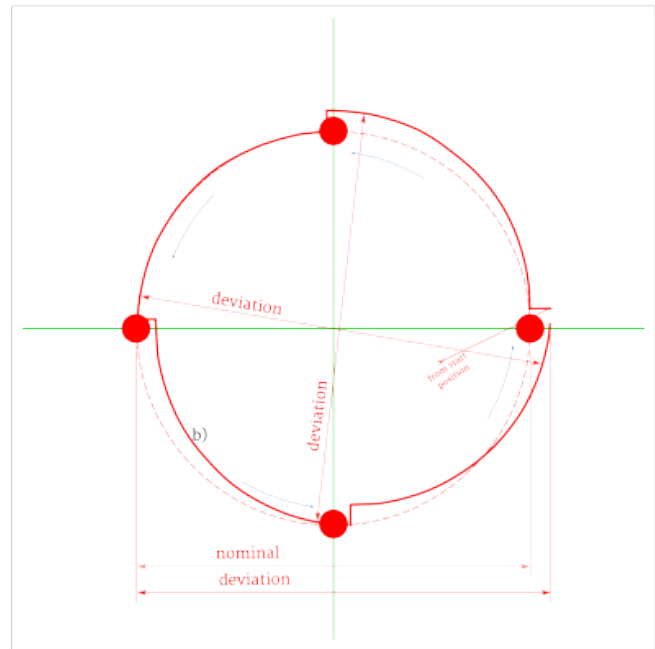
For either movement it is assumed that in the starting position (lower left) both belts are fully in the end stop of the pulley.

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.



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

The RepRap Industrial 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 RepRap Industrial 3D printer can be found in the [Service guide](#).



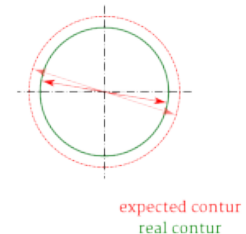
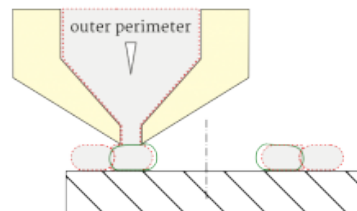
Effective travel of the print head when backlash is over-compensated.

Small holes

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

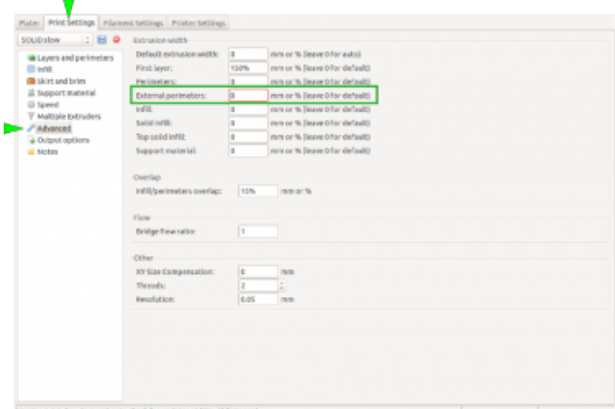
When the print head follows a curve the inner side of an extrusion strand is slightly overfilled which results in an excess of material on the inner wall of a hole.

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.



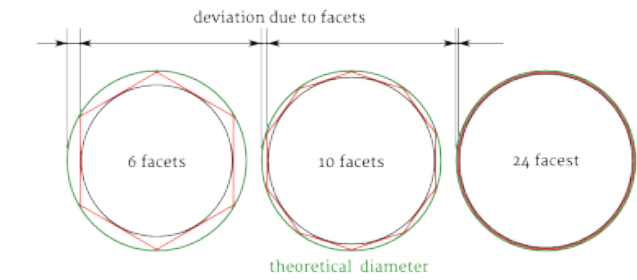
Reducing the outer perimeter extrusion width in Slic3r.

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.