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.

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

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.

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 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.

3D printer preparation

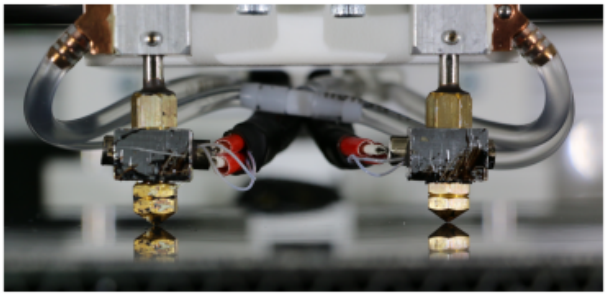
Depending on the task, the setup of the 3D printer for dual extruder prints may vary.
Printing bicolored objects will mostly be done with the same resolution requirements throughout the print, so installing the same nozzle diameter on both hot ends is recommended.
For printing support structures with 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.

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.

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.

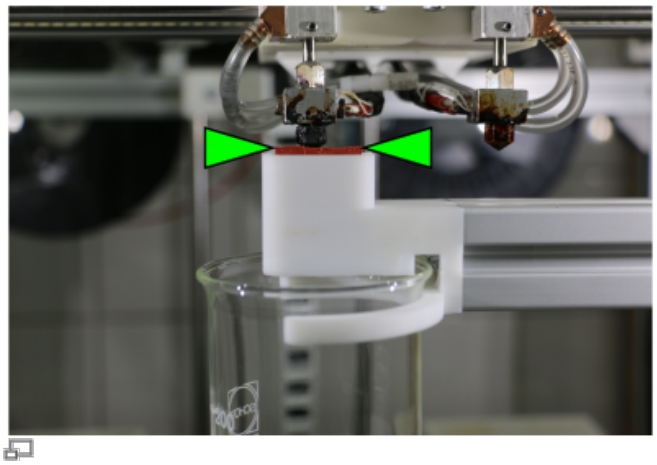
The Wiper Upgrade Kit provides the necessary hardware for successfully printing dual extruder print jobs.

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.

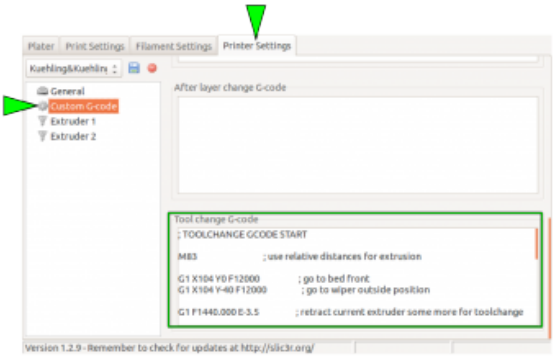
INFO

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.

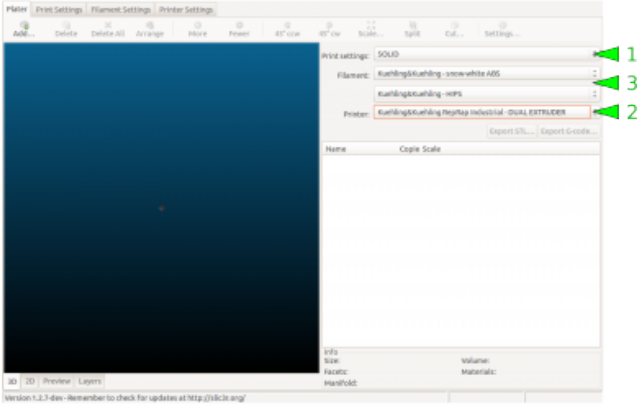


The Slic3r tool change G-code provides the information for the wiping process. It is preconfigured and normally does not need interference.

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:



Preselecting in Slic3r for dual extruder printing.

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.

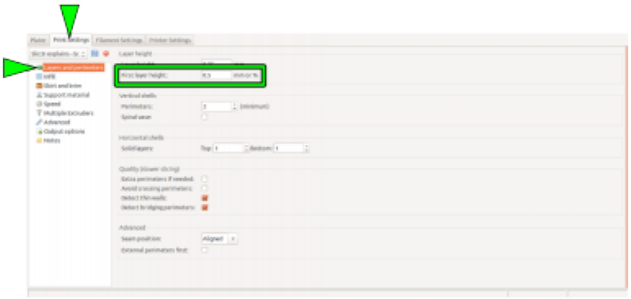
Menu “Print Settings”

→ 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.



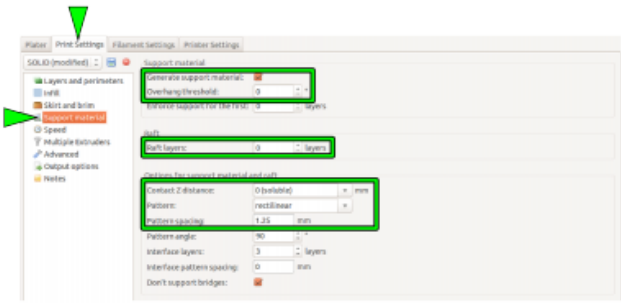
First layer height and support material extruder nozzle diameter must correspond.

→ 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*).

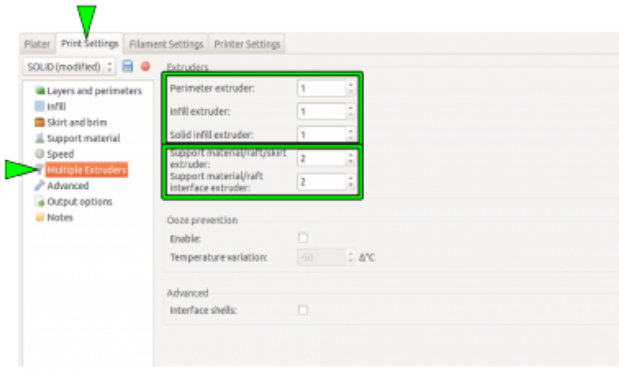
- The `Contact Z distance` must be “0 (soluble)” with a (recommended) “rectilinear” `Pattern` and a `Pattern spacing` of 1.25 mm.
- It is a good idea to generate 5 - 7 `Raft` layers with the support material. Thus, slight leveling mistakes are compensated and an even and reliable subsurface is always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

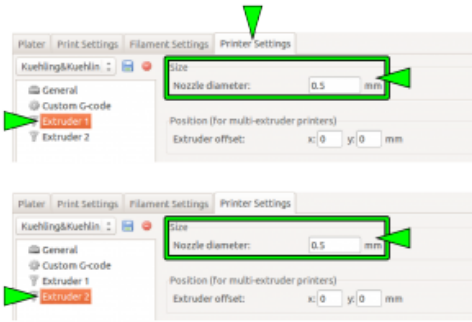


Assigning the hot ends to their respective tasks.

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.

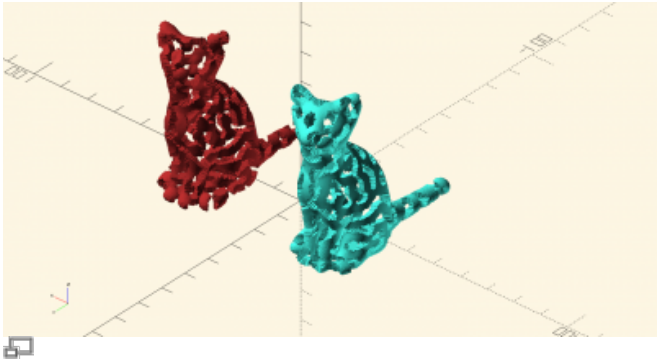
DANGER OF INJURY!

d-limonene is a solving agent and therefor 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.

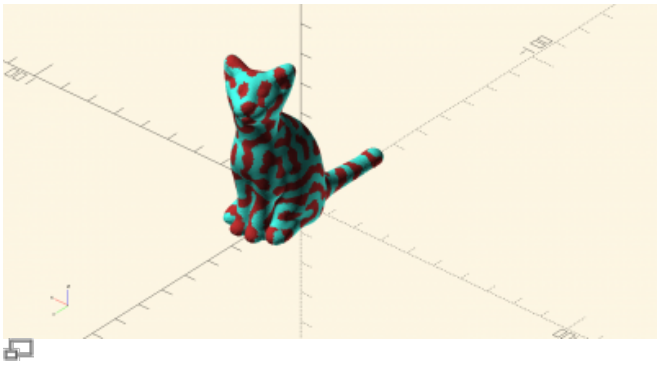
After a complex ABS model has successfully been printed with HIPS soluble support material, the support material must be removed and dissolved. This is an easy process that only requires only a few steps:

- 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 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.

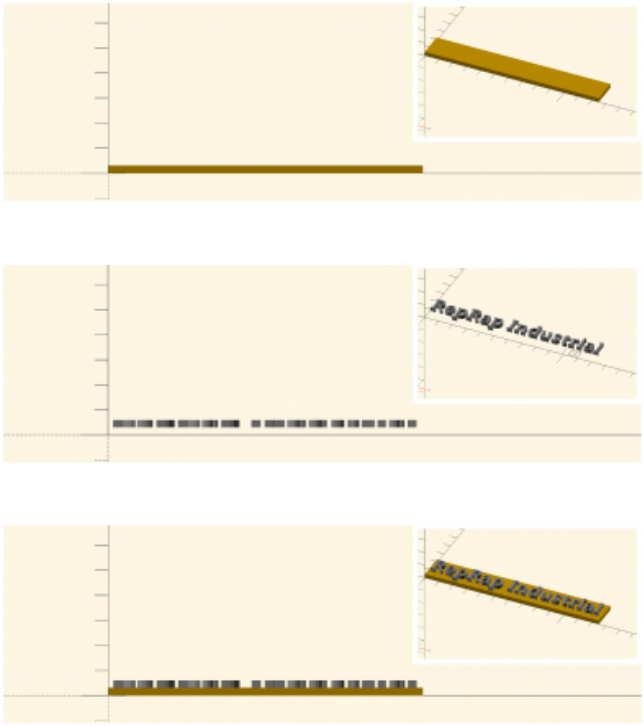
Bicolored printing



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.



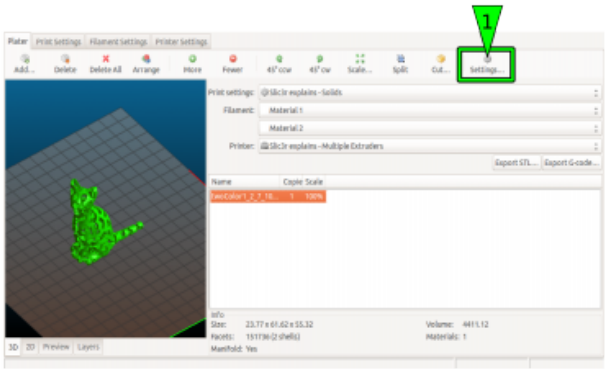
3D model preparation for bicolored printing example 2: both individual parts of the 3D model being aligned in a common assembly and exported as separate STL-files with the correct relative coordinates.

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 therefor 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:

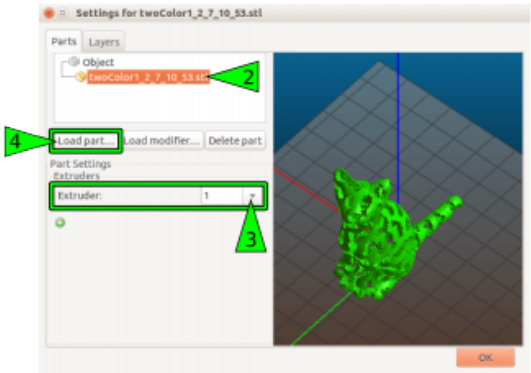
- Upload the first part to the *Plater*.
- Open the `Settings` dialog.



The first part is loaded to the Plater directly.

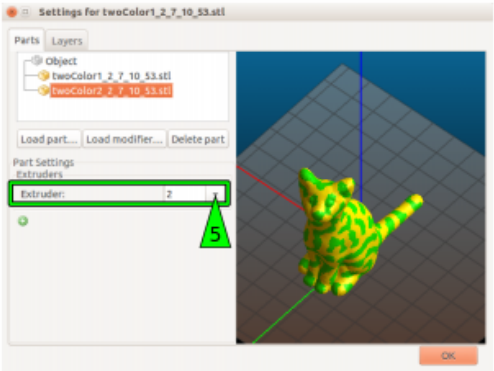
- Mark part one and assign it to extruder 1.
- Click `Load part …` and add the second

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



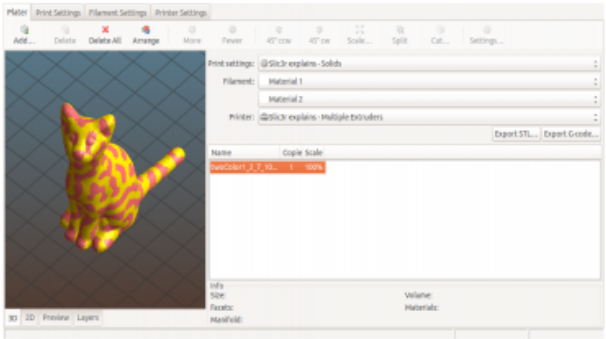
After the extruder for the first part has been assigned, the second part is loaded via the *Settings* dialog.

- 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:

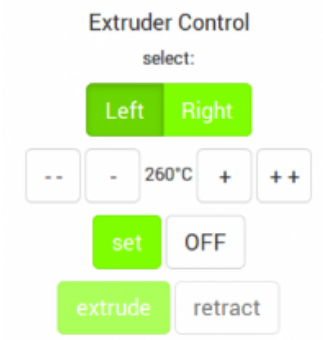
- Determining material properties
  - Material specific extrusion temperature
  - Finding the extrusion multiplier
  - Filament quality
- 3D-suitable construction and slicing
  - Handling 3D-files
  - Layers and quality
  - Dimensional accuracy

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 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.

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