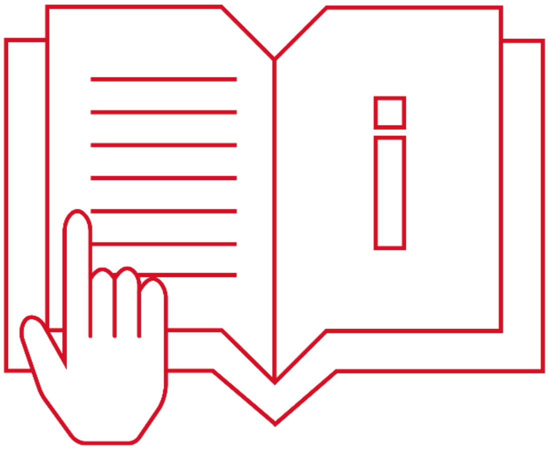
GUIDE FOR BASIC EDITION OF MATERIAL PROFILES IN 3DGENCE SLICER



Spis treści

1. INTRODUCTION

3DGence Slicer has been prepared for 3DGence printers with ready-made print settings for dedicated materials. Each of the profiles available in 3DGence Slicer was developed by a team of specialists and then tested for over 1000 hours on many complicated test models. Therefore, at the beginning of the adventures with 3D printing, we recommend using default profiles. The manufacturer guarantees the highest possible quality of models made using dedicated software and materials as well as pre-set default settings.

For advanced users, an option to change print settings has been made available. This option is designed to modify existing profiles to get quality improvement on materials not certified by the manufacturer or adjusting the device's operating settings. Due to the nature of parameter modification, the manufacturer does not guarantee the quality and repeatability of prints prepared in this way.

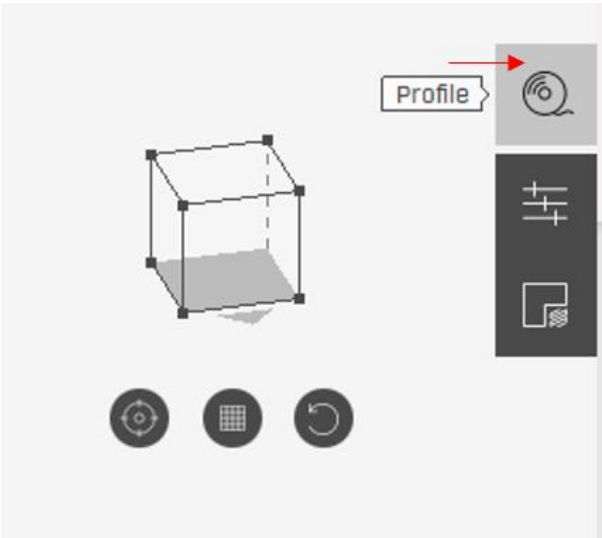
Please note that the use of materials and print settings not provided by the manufacturer is the responsibility of the customer. This means that the manufacturer does not provide support for the use of advanced profiles.

3DGence Slicer uses the modified CuraEngine engine while maintaining parameter names, also experienced users can use the extensive documentation of the Cura software community. The names and functions of the functions are identical with the Cura software.

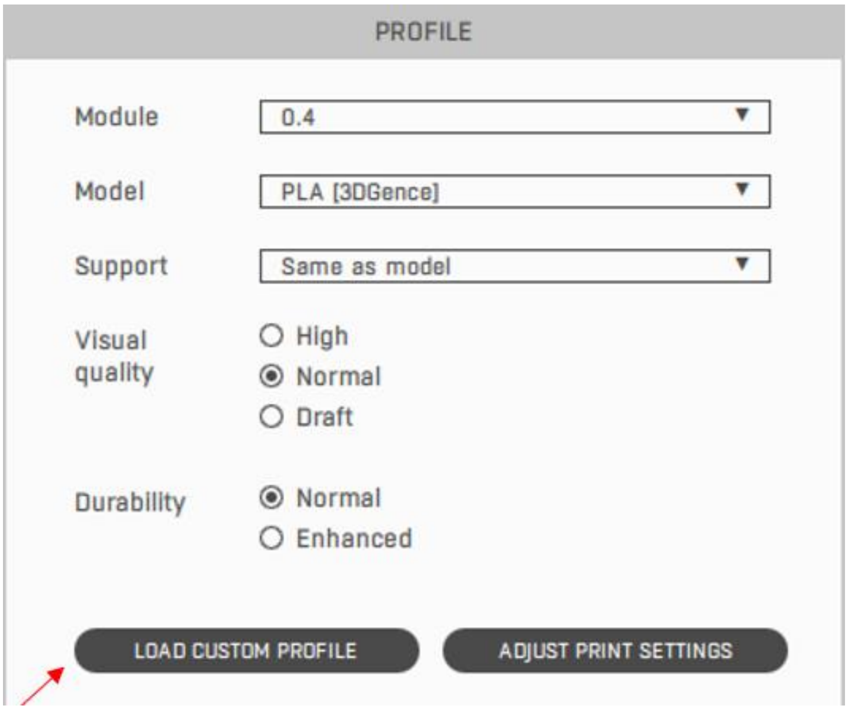
2. EDITION OF ADVANCED OPTIONS

The following is a brief guide to the basic edition of dedicated advanced profiles.

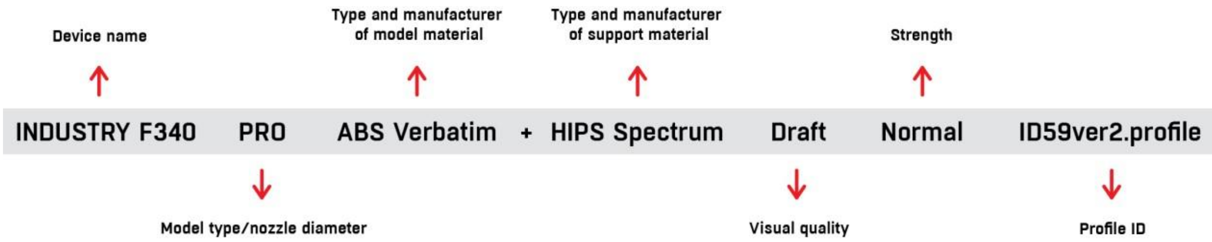
1. Select the "PROFILE" menu on the right.

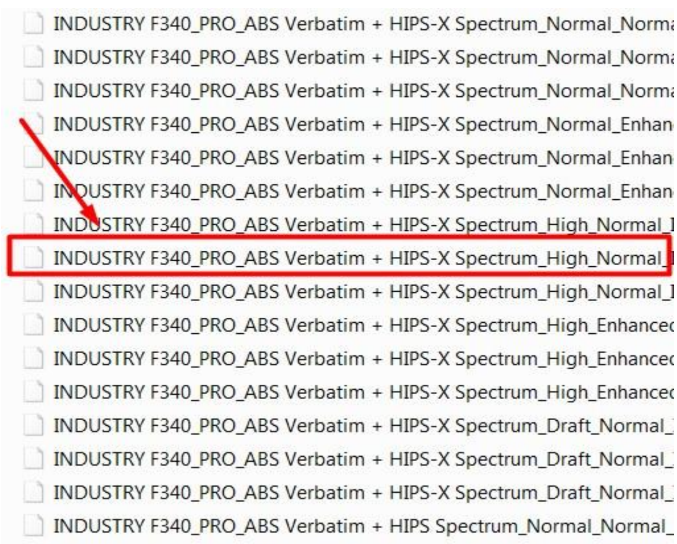


2. Select the “LOAD custom profile” option.

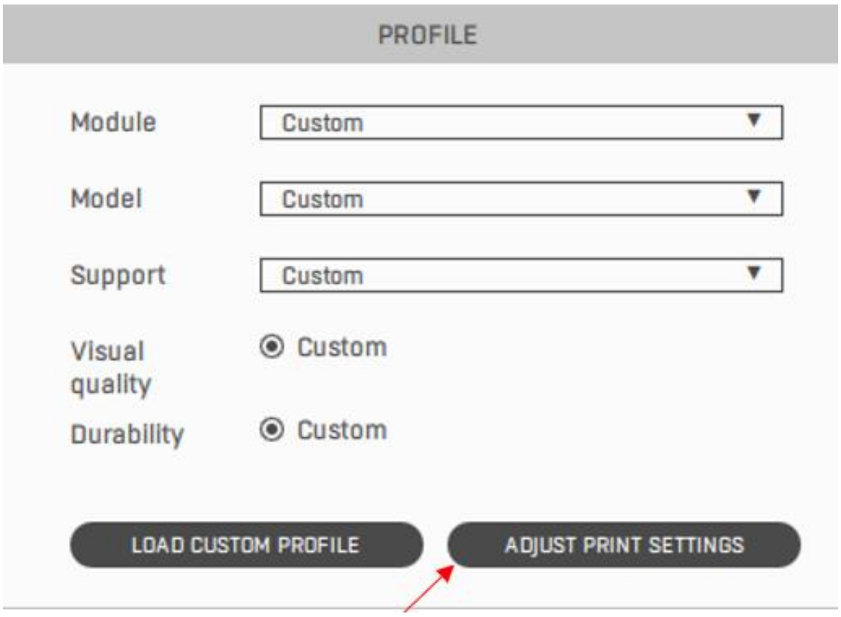


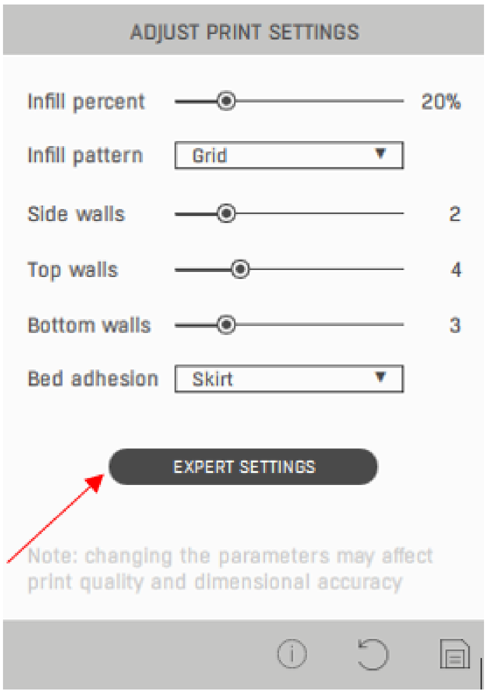
3. Select the profile you want to edit from the list, then select "Open" (it is recommended to select the profile with the latest update date).





4. After loading the profile in the "PROFILES" menu, you will be able to see the change of the profile name to "Custom". To go to the advanced edition select ADJUST PRINT SETTINGS and then select EXPERT SETTINGS.





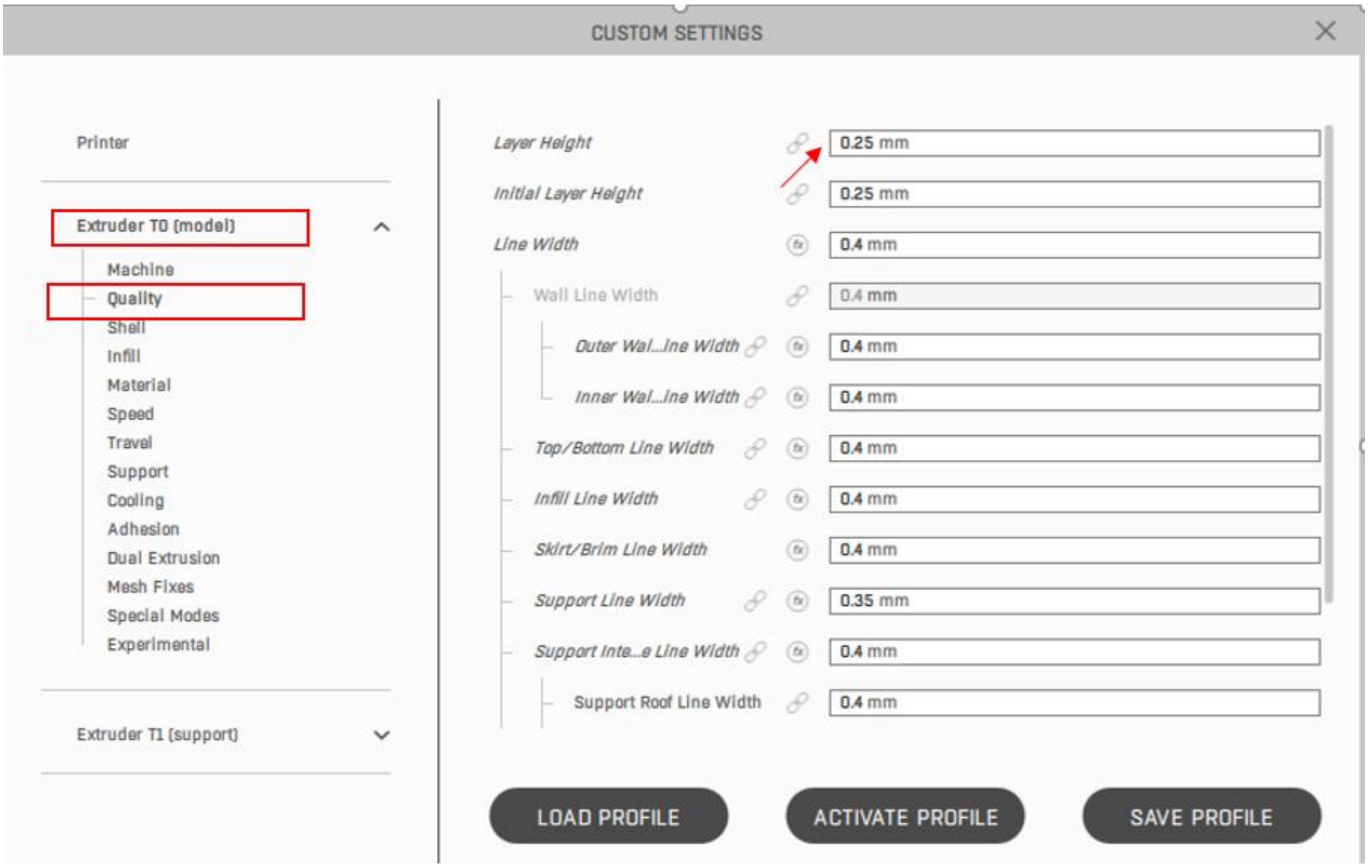
ATTENTION!

It should be remembered that changes to these settings may require adjustment of other parameters. The following are the basic changes that should be applied until the desired results are obtained.

As follows from the observations - the most common changes of the default profiles concern changes in the layer height and density of the infill.
The lower layer height guarantees a more accurate mapping of the model in reality. The print time will increase as the layer height decreases (assuming that we leave the remaining parameters unchanged). However, increasing the layer height reduces the time of printing at the expense of the visual effect. It is assumed that the value of the layer height should not exceed the diameter of the nozzle. The infilling of the model is mainly responsible for the durability of the print. In contrast to the side walls and the bottom and top layers of the model, the infilling can be of different density (from 0 - 100%) the more densely we generate the infilling, the more durable the model will be, and its mass and print time will increase.

2.1. Changing the height of the layer

To change the layer height, select EXTRUDER T0 (model) → QUALITY-> LAYER HIGH in the advanced settings and then enter the new layer height value in the LAYER HEIGHT window.
ATTENTION: If the Layer Height will be change on Extruder T0, the value on Extruder T1 will change automatically on the same value.

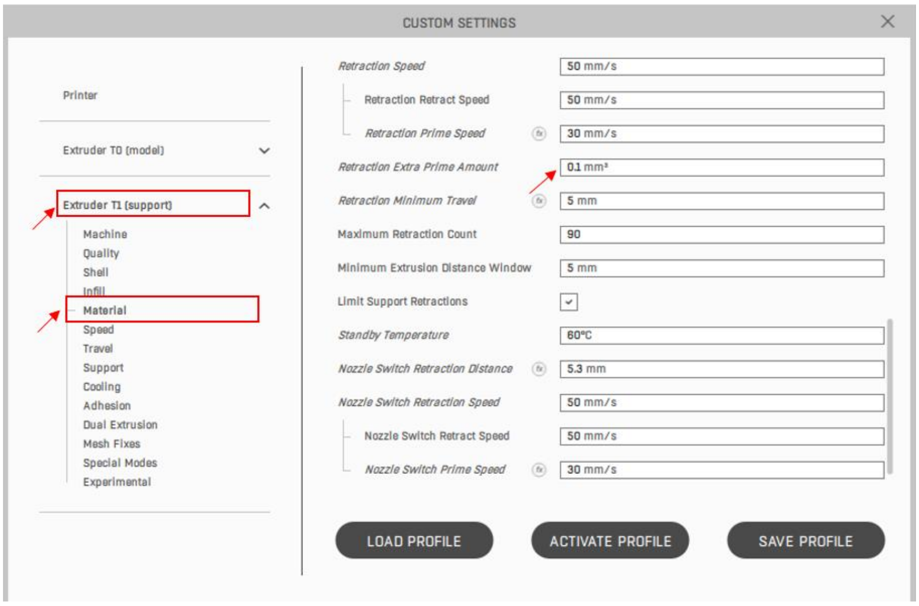


When changing the height of the layer from larger to smaller (e.g. from 0.25mm to 0.2mm), pay special attention to:

- Correction of retraction settings:

1st Extruder - Material - Retraction Extra Prime Amount.

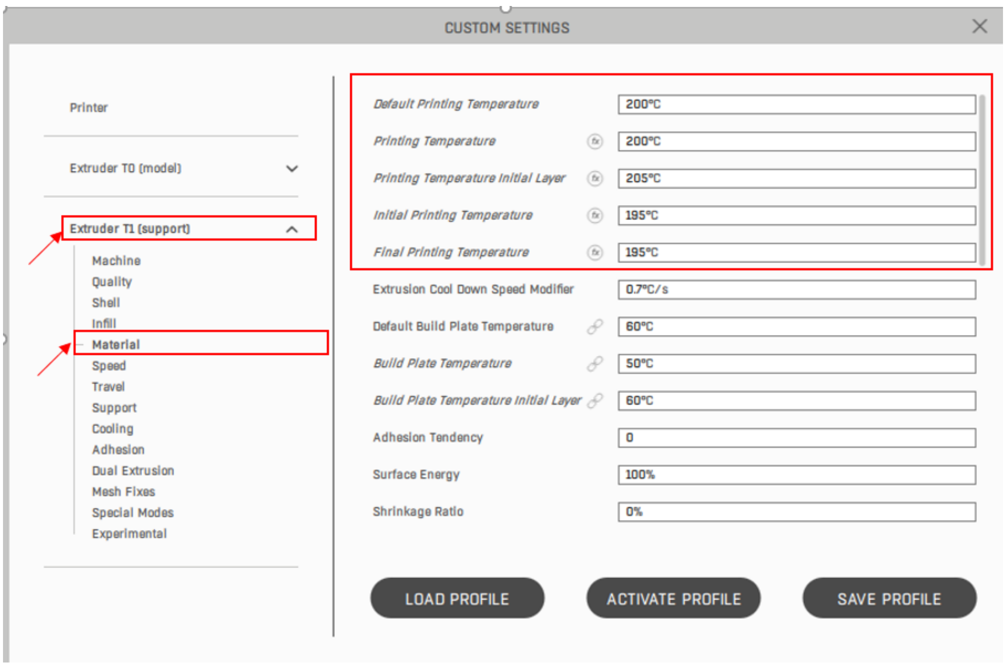
If on the printed model you notice material pouring after retraction, the value in the field Retraction Extra Prime Amount should be gradually decreased by 0.01mm3 until the desired result is obtained.



- Correction of all hotend temperatures

1st Extruder - Material - Default Printing Temperature/Printing Temperature/Printing Temperature Initial Layer/Initial Printing Temperature.

It is recommended to reduce the temperature by 2°C. By reducing the layer height, we change the flow of the filament in time, a slight lowering of the temperature gently increases the resistance of pressing the material, which results in reduced inertia and improved printing stability. The values should be reduced until the desired result is obtained.

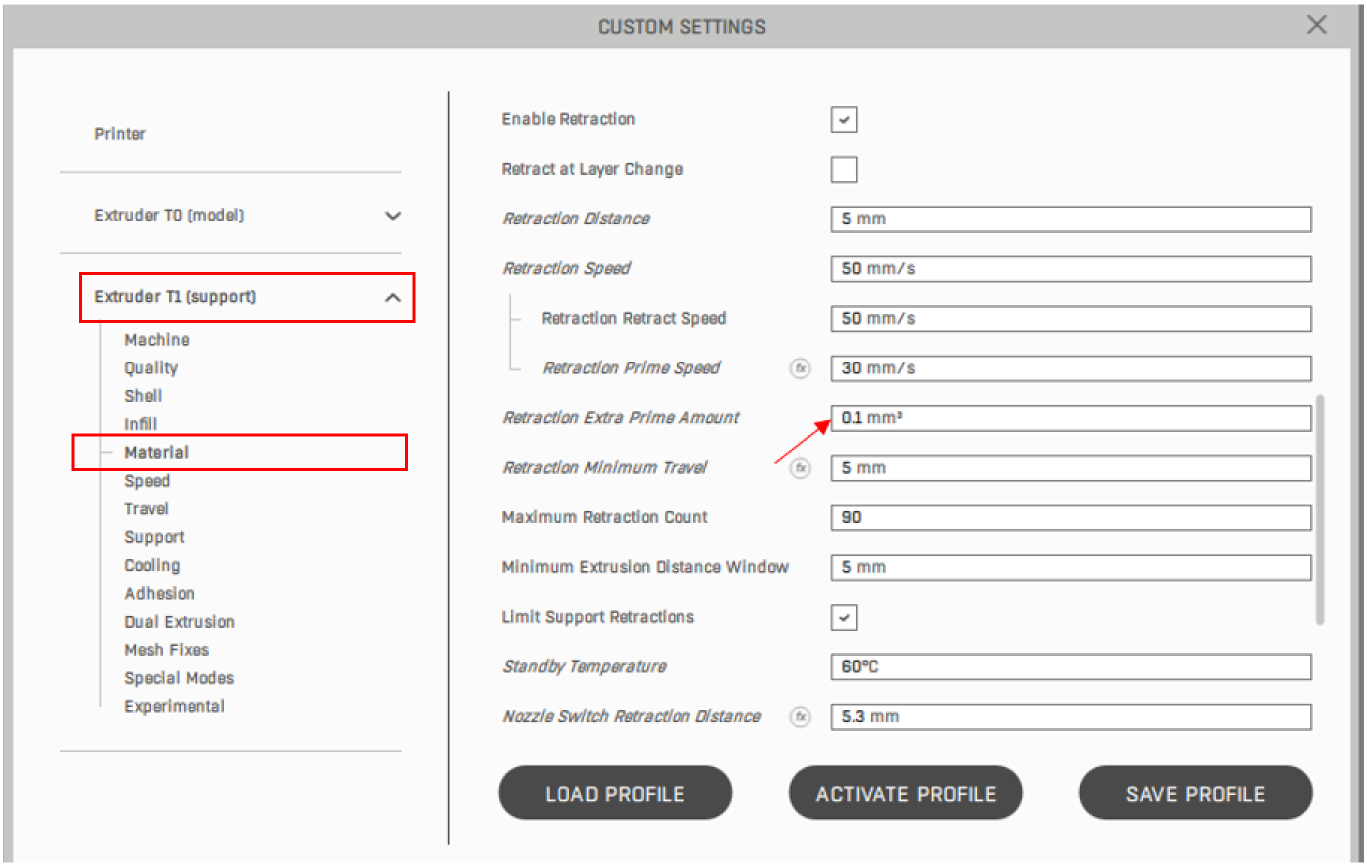


When changing the height of the layer from smaller to greater (e.g. from 0.15mm to 0.2mm), pay special attention to:

- Correction of retraction settings:

1st Extruder - Material - Retraction Extra Prime Amount.

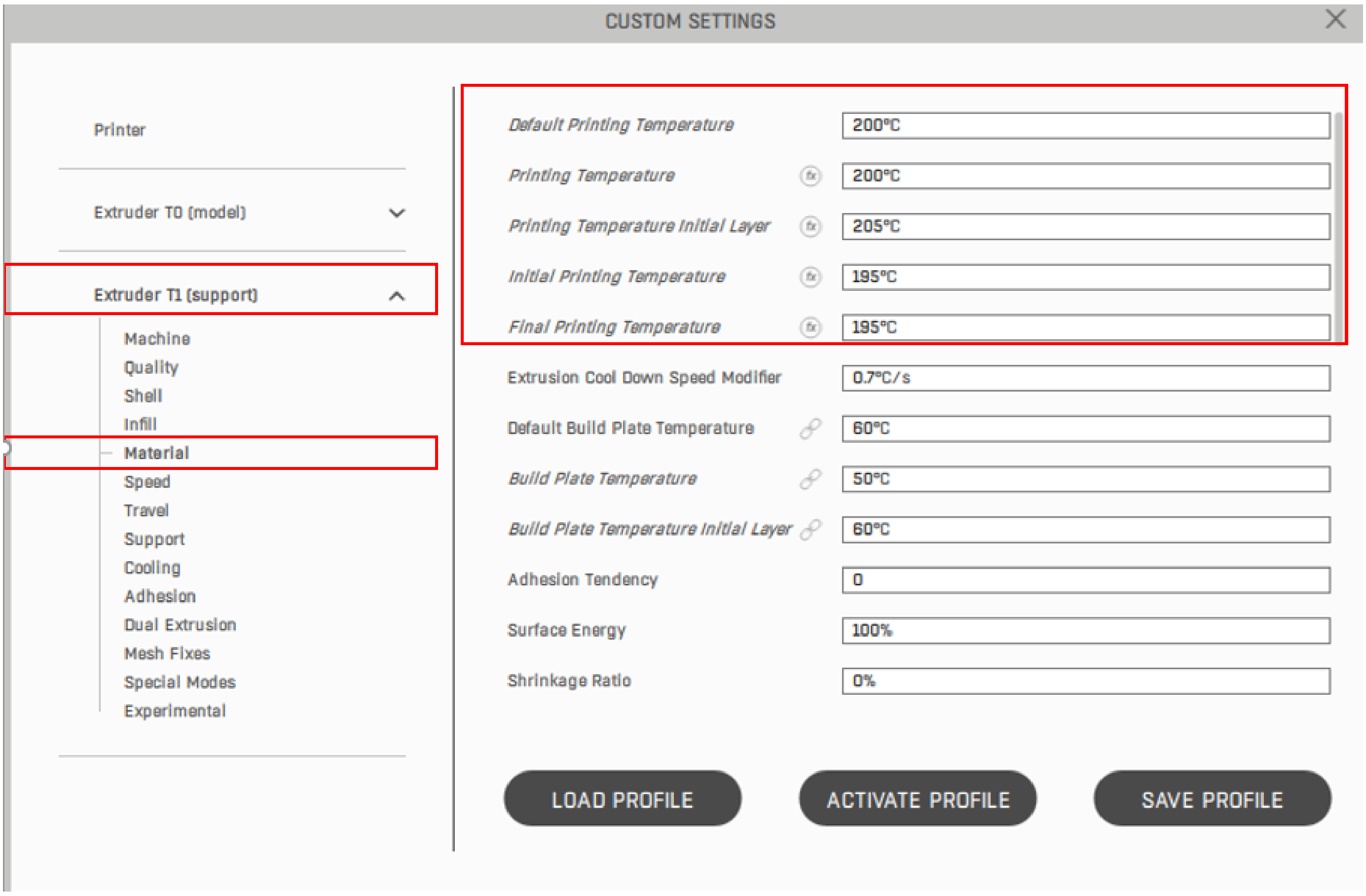
If on the printed model you notice that the material is not in perfect condition after retraction, the value in the field Retraction Extra Prime Amount should be increased gradually by 0.01mm3 until the desired result is obtained.



- Correction of all hotends temperatures:

1st Extruder - Material - Default Printing Temperature/Printing Temperature/Printing Temperature Initial Layer/Initial Printing Temperature.

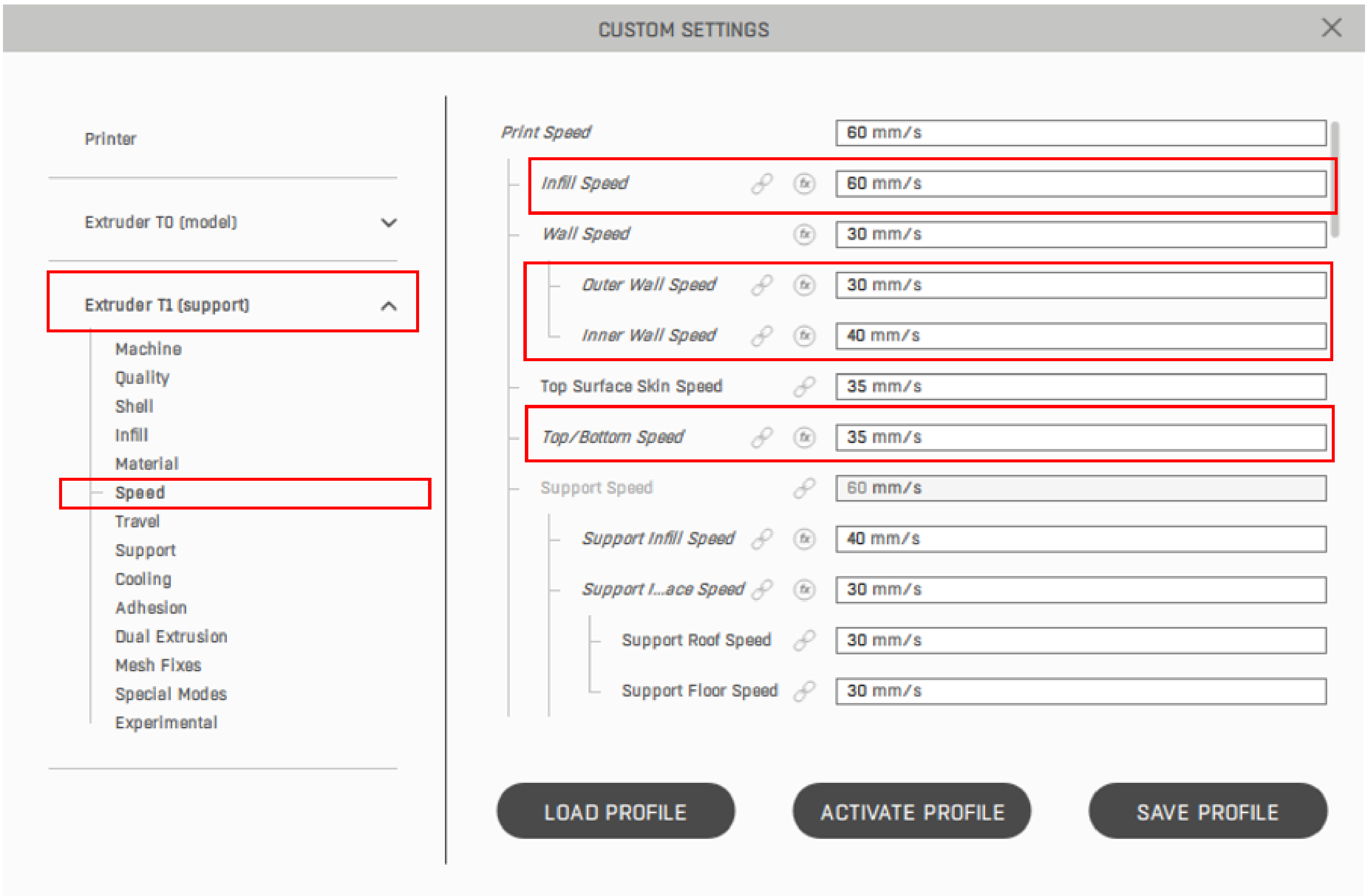
It is recommended to increase the temperature gradually by 2 ° C. By increasing the height of the layer, we increase the flow of the filament in time, a slight increase in temperature slightly reduces the resistance of the material, which results in thinning of the material and its more accurate feeding. Values should be increased until the desired result is obtained.



- Correction of all printing speeds:

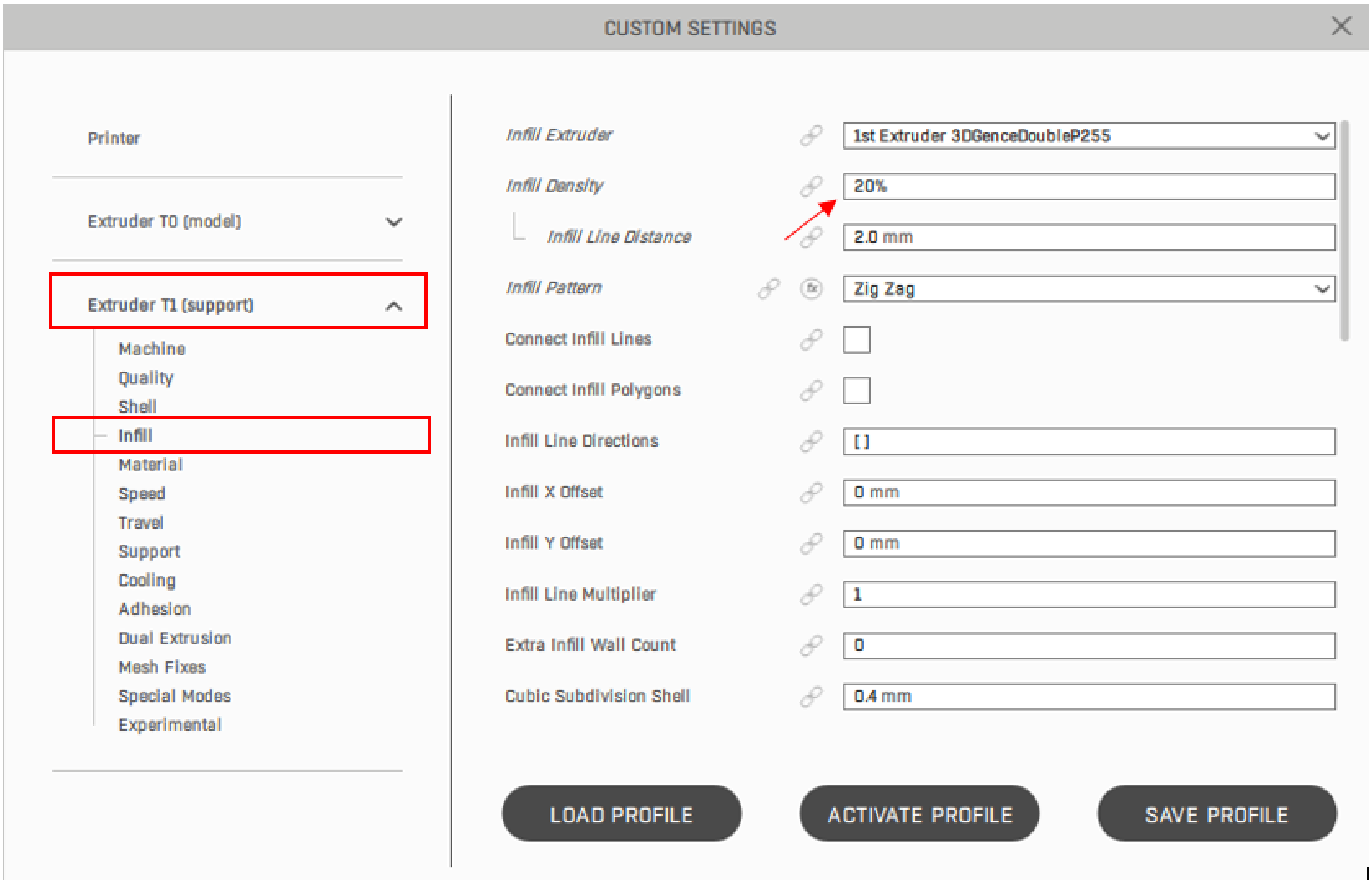
1st Extruder - Speed - Infil Speed/Outer Wall Speed/Inner Wall Speed/Top/Bottom Speed.

It is recommended to reduce the speed iteratively by 3 mm/s. A slight reduction in velocity allows to eliminate many errors resulting from a higher layer height, as well as to reduce the effect of "ghosting", i.e. rounding of walls, which can be more visible at a higher layer.



2.2. Changing the density of the infill

To change the infilling density, one must first select 1ST EXTRUDER → INFILL in the advanced settings and then enter the new infill density value in the INFILL DENSITY window.



When changing the infilling density from smaller to larger (eg from 20% to 40%), particular attention should be paid to:

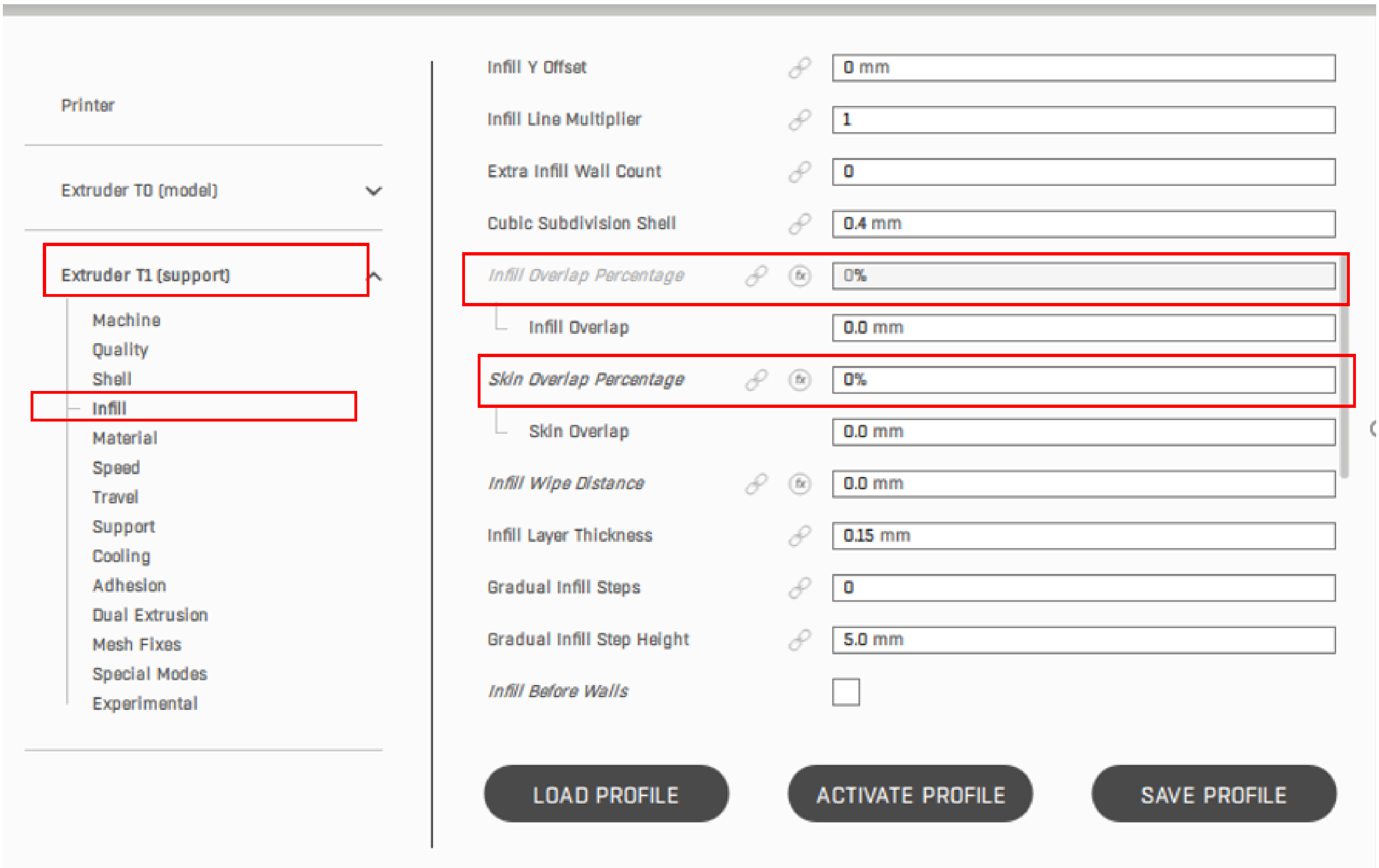
- Correction of infill settings:

1st Extruder - Infill - Infill Overlap Percentage.

If on the printout we notice the material collapse at the bonding points of the model with the walls and the value of the Infill Overlap Percentage parameter is greater than 0, reduce this value by 2% until the desired result is obtained.

1st Extruder - Infill - Skin Overlap Percentage.

If on the printout we notice the material stacking up at the points of joining the upper walls with the model walls and the value of the Skin Overlap Percentage parameter is greater than 0, this value should be decreased by 2% until the desired result is obtained.



When changing the infilling density from larger to smaller (eg from 40% to 20%), pay special attention to:

- Correction of the number of upper and lower layers:

1st Extruder - Shell - Top Layers/Bottom Layers.

Decreasing the infilling it is worth increasing the number of upper and lower layers. This applies in particular to the case when we assume that the infilling is not needed or we only want to minimally strengthen the inside of the printed body using, for example, a 5% infill. Increasing the upper layers is important in the case of such a thin infill, because often on such a infilling we will have to build a flat surface.