

# **USING YSOFT BE3D DEECONTROL**

# CONTENT

<b>1</b>	<b>Introduction</b>	<b>3</b>
<b>2</b>	<b>What is DeeControl?</b>	<b>4</b>
<b>3</b>	<b>DeeControl requirements</b>	<b>5</b>
<b>4</b>	<b>Installation of DeeControl</b>	<b>6</b>
4.1	Windows installation	6
4.2	Mac installation	9
<b>5</b>	<b>First setting of DeeControl</b>	<b>11</b>
<b>6</b>	<b>Print job preparation</b>	<b>12</b>
<b>7</b>	<b>Print settings</b>	<b>15</b>
7.1	Simple settings	15
7.2	Advanced settings	16
<b>8</b>	<b>Saving or sending the print job to SafeQ</b>	<b>20</b>
<b>9</b>	<b>GCode preview</b>	<b>22</b>

# 1 INTRODUCTION

The purpose of this document is to provide Y Soft end user with a complete overview how to install the program and how to best prepare the print job for printing.

## 2 WHAT IS DEECONTROL?

The DeeControl application is a slicer software for 3D jobs, it converts a 3D model into printing instructions for your 3D printer. It cuts the model into horizontal slices (layers), generates tool paths to fill them and calculates the amount of material to be extruded.

## 3 DEECONTROL REQUIREMENTS

### Software:

- Operating system: Windows 7, Windows 8, Windows 8.1, Windows 10, MAC OS X 10.7 and later
- DirectX 10 or higher (OpenGL)

### Hardware:

- Intel Core i3 processor or a processor with similar performance
- 2 GB RAM
- 2 GB hard disk space

## 4 INSTALLATION OF DEECONTROL

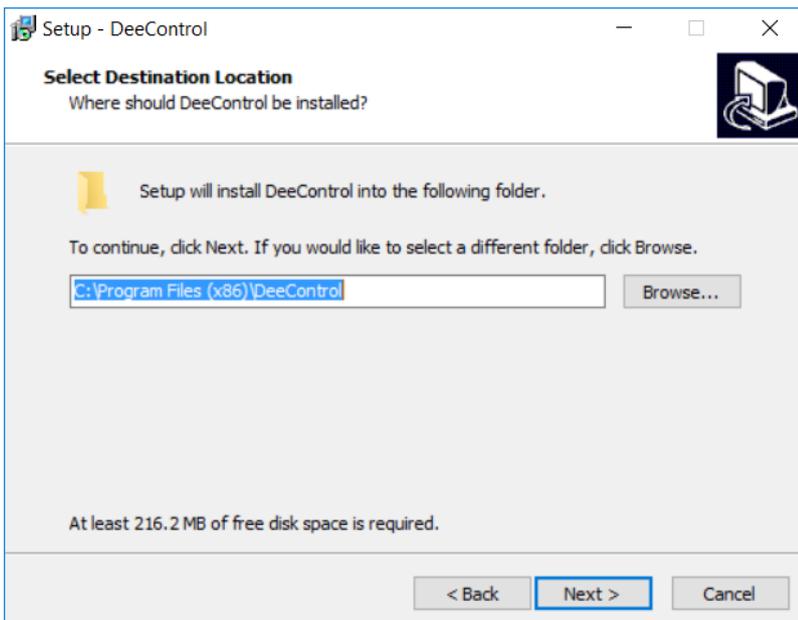
Ask your administrator for DeeControl installation file.

### 4.1 WINDOWS INSTALLATION

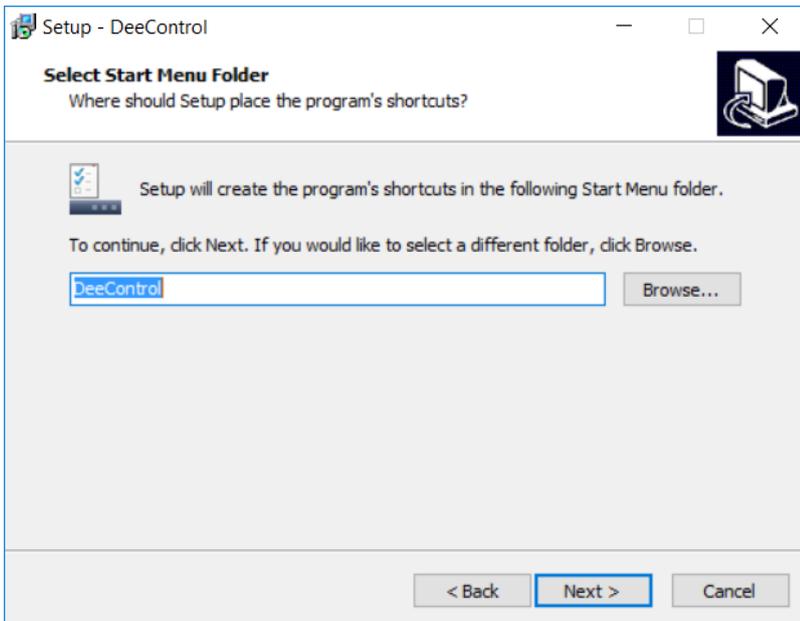
1. Locate the file on your workstation
2. Run the installation file and follow the wizard:



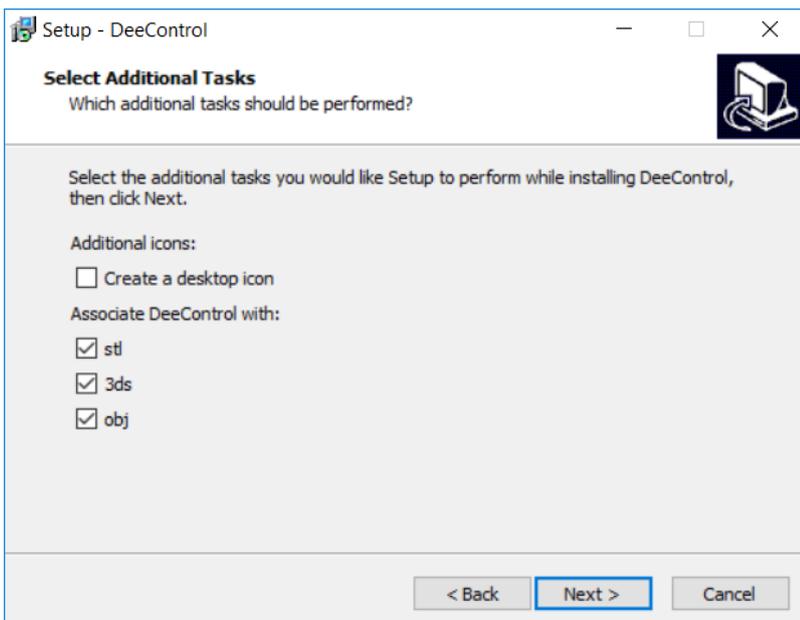
- The installation directory can be changed during setup.



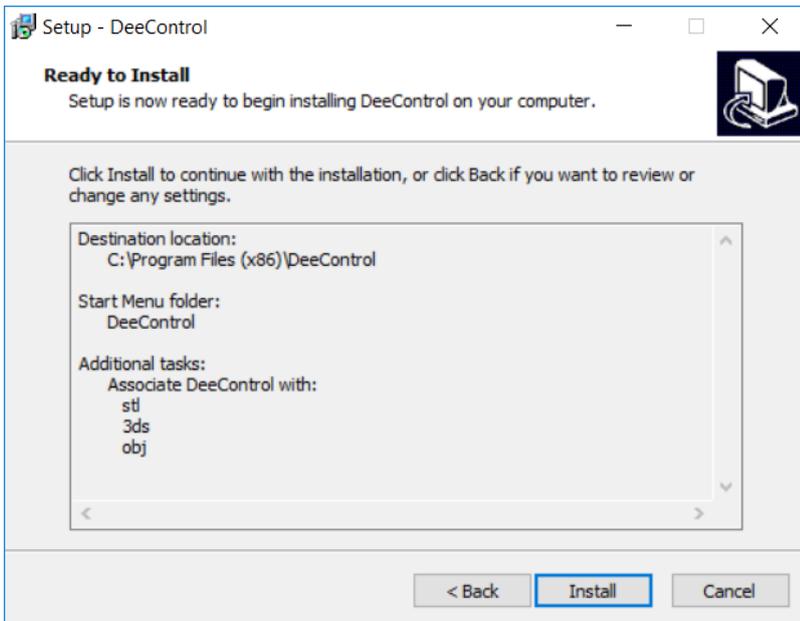
- Start menu folder can be changed during the installation



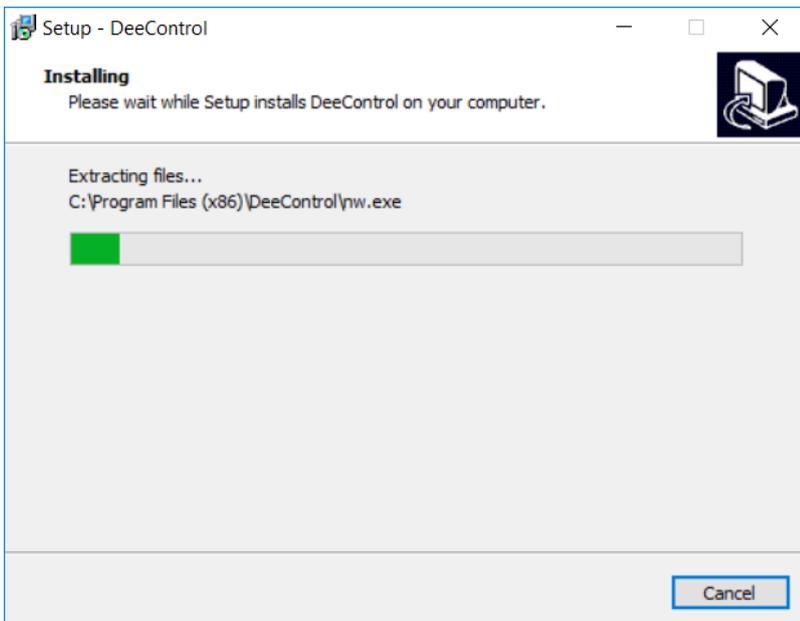
- The user may enable supported file type association during setup. If enabled, the given file type will always be opened with DeeControl. Desktop icon can be created.



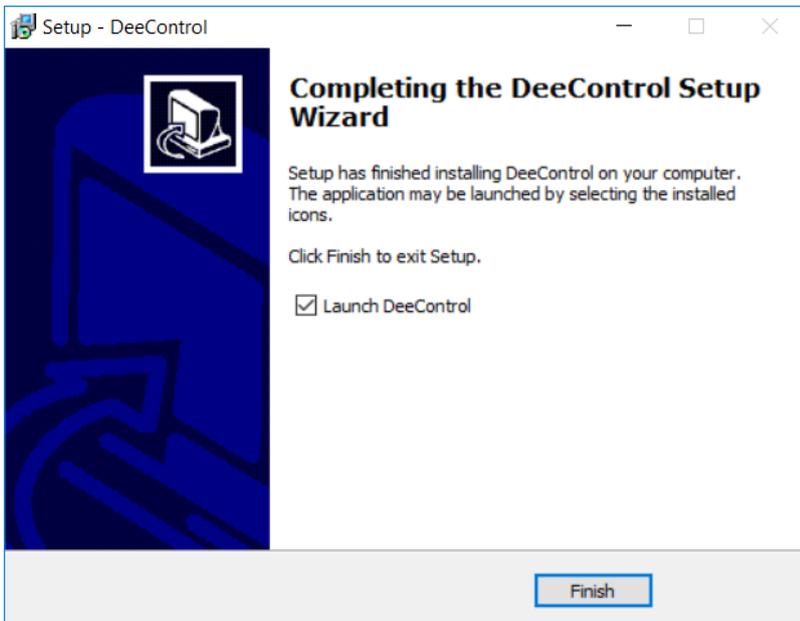
- Once all the selections are made, the installation starts upon selection of Install



- The progress is displayed during the installation



- Once the installation is successful a confirmation message is displayed



## 4.2 MAC INSTALLATION

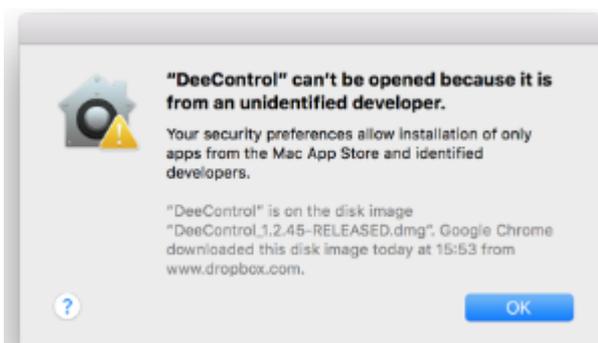
Once the .dmg file is opened the application icon is displayed



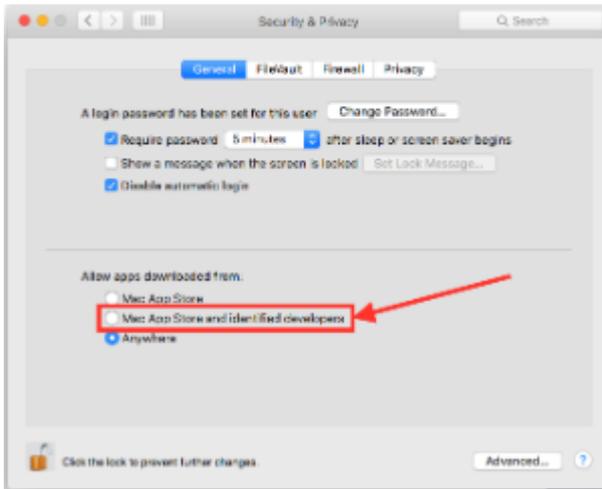
User should Drag&Drop the icon to his applications to gain access it conveniently. Otherwise it can be used directly from the stored location.

In some cases (based on your Mac security settings) the MAC OS [gatekeeper service](#) recognizes the SW as unidentified and therefore some customers won't be able to install it due to their company's security restrictions .

Following error message is shown:

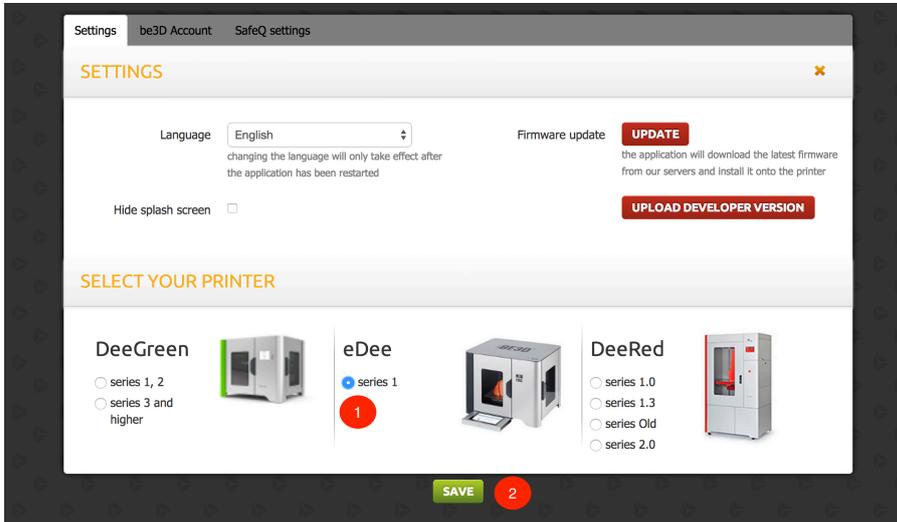


In case company security restrictions allow changes, go to **System preferences > Security&Privacy** - enable everywhere



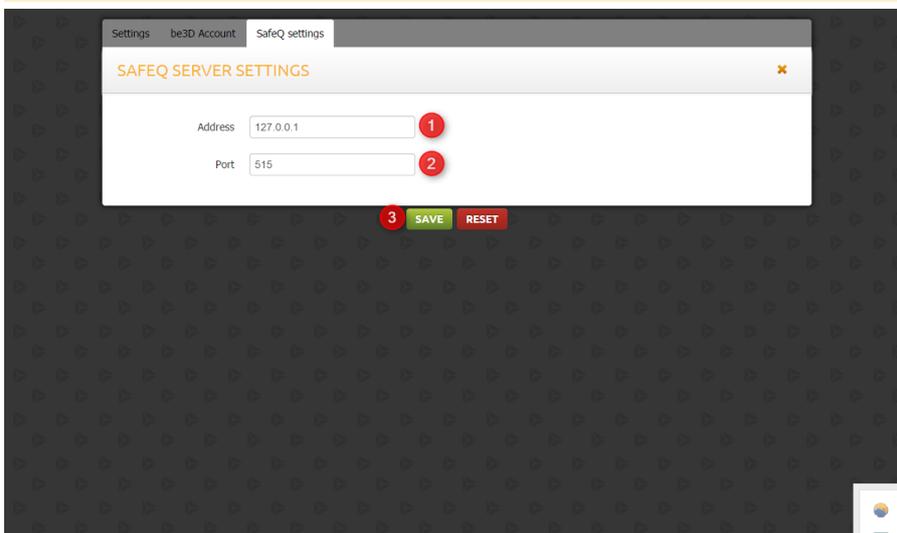
## 5 FIRST SETTING OF DEECONTROL

1. After the first run of DeeControl open **Settings** menu.
2. Select eDee and **SAVE** settings.



3. Open the **Settings** window again and enter the **SafeQ server** settings and type IP address of Flexispooler and Port (default value is 515)

In case you are using Flexispooler on your computer the IP address is always 127.0.0.1 - discuss this with your administrator.



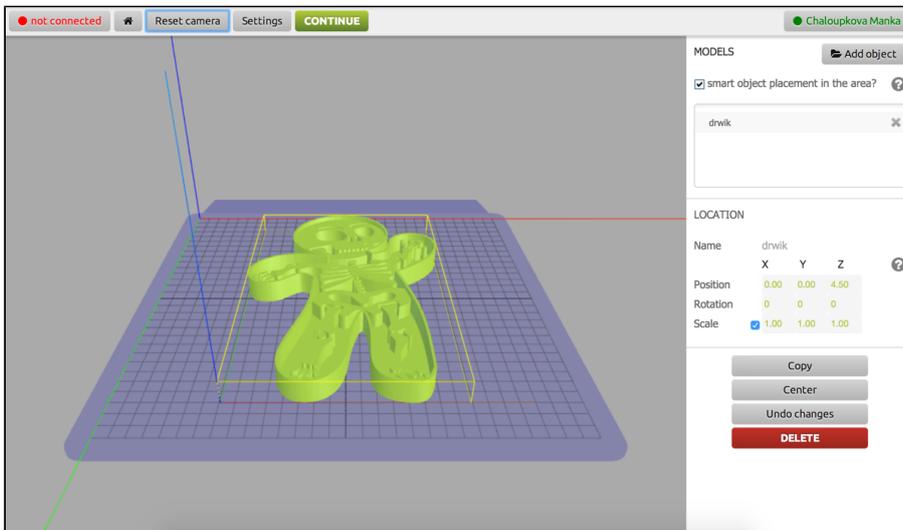
## 6 PRINT JOB PREPARATION

a) A 3D print job begins with a 3D digital model. The supported file types are:

- STL
- OBJ
- 3DS

b) Open DeeControl on the client workstation.

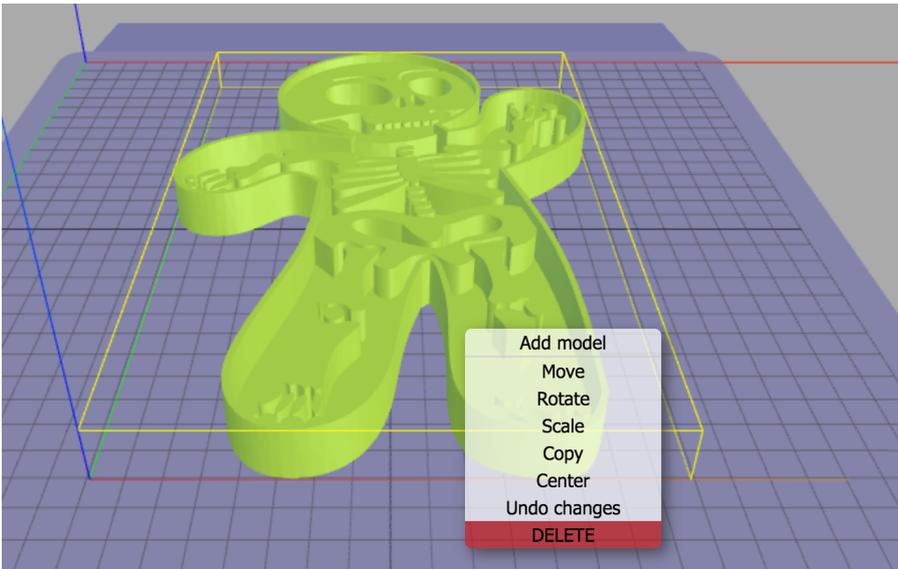
c) Load your 3D model file and prepare it on the DeeControl's virtual print bed.



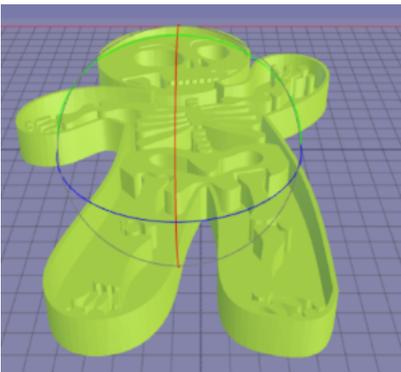
You can do following operations with the model via the location section in the right side panel:

- **Move** - change the X,Y,Z coordinates of the object center point
- **Rotate** - rotate the object in X,Y,Z axis
- **Resize** - you can resize the object proportionally with the checked checkbox in front of the values, or you can deselect it and change only the desired dimension

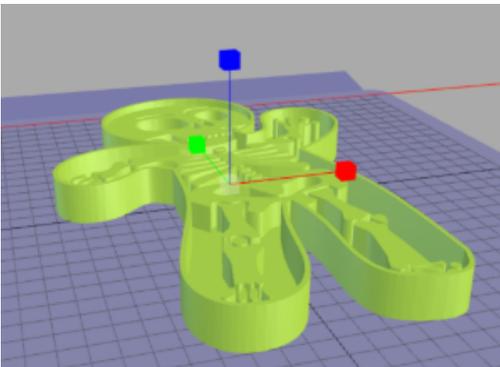
Or you can do a mouse right-click on the object and a menu with appropriate actions appears



- **Add model** - you can add additional model to the bed
- **Move** - you can just Drag&Drop the object to the desired location
- **Rotate** - you can rotate the object around desired axis: click&hold on the desired axis, move to rotate the object.



- **Scale** - you can manually scale the desired dimension: select the axis and then use Drag&Drop technique to put it desired shape



- **Copy** - you can multiply the object on the print bed
- **Center** - aligns the object in the middle of the print bed

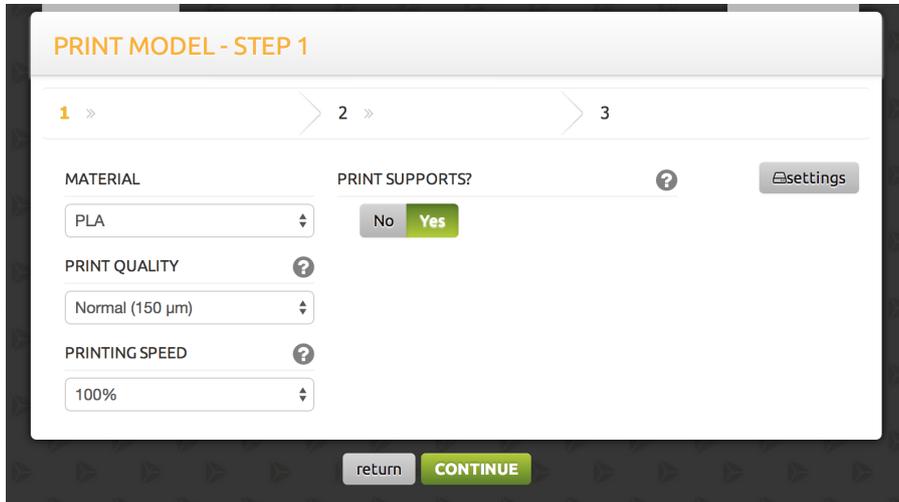
- **Undo changes** - removes any operation performed on the object

d) When the model is positioned as desired on the virtual print bed, select Continue

# 7 PRINT SETTINGS

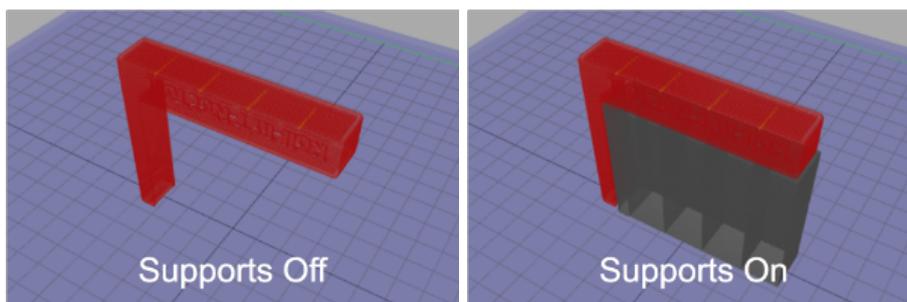
## 7.1 SIMPLE SETTINGS

Once the model is in desired place, in desired size and shape, the setting window is displayed:



Following simple options are available:

- **Material selection** - list of recommended and approved materials to be used with the printer, by selecting the right material some advanced settings are automatically used for the printing (such as optimal temperature, speed, layer thickness, etc.)
- **Print Quality** - this option determines the layer height of the print. Using lower value result in finer model but the print time will be significantly longer.
- **Printing speed** - printing speed might be an important parameter when one wants to achieve a high level of details. Fast printing doesn't give you nice level of details, but it shortens the printing time, which is suitable for fast drafts in the first phase of prototyping or when details are not important. Slow printing usually provides higher level of details, but it takes significantly more time, but this is usually acceptable when printing the final product.
- **Supports** - the option to enable generation and printing of supports where necessary (defined by the slicer engine), Using supports will enable you to print models with overhangs. The supports are not generated for models that do not require them.



## 7.2 ADVANCED SETTINGS

The advanced settings are suitable only for highly experienced users and has to be used wisely! E.g. student of technical university.

## ADVANCED SETTINGS



The selected values will replace the original settings and will be valid until reset.

?	Shell thickness (mm)	0.4		4	<b>0.8 mm</b>
?	Top solid layers	0		20	<b>6</b>
?	Bottom solid layers	0		20	<b>3</b>
?	Infill density (%)	0		100	<b>18%</b>
?	Support type	lines			<b>lines</b>
?	Angle of support creation (°)	25		60	<b>30°</b>
?	Support density (%)	10		100	<b>16%</b>
?	Support everywhere	Yes			<b>Yes</b>
?	Initial structure	none			<b>none</b>
?	Raft density (%)	10		100	<b>25%</b>
?	First layer air gap (mm)	0		1	<b>0.28mm</b>
?	Support printing speed (mm/s)	10		90	<b>70mm/s</b>
?	Infill printing speed (mm/s)	5		90	<b>80mm/s</b>
?	Use fan	Yes			<b>Yes</b>
?	Outer perimeter printing speed (mm/s)	5		90	<b>40mm/s</b>
?	Inner perimeter printing speed (mm/s)	5		90	<b>50mm/s</b>
?	Solid layer printing speed (mm/s)	5		90	<b>40mm/s</b>
?	Temperature of first nozzle (°C)	150		255	<b>205°C</b>
?	Turn on bed heating	Yes			<b>Yes</b>
?	Bed temperature (°C)	50		110	<b>55°C</b>
?	Chamber temperature (°C)	10		70	<b>10°C</b>
?	Axis Z offset during extruder exchange (mm)	0		10	<b>0mm</b>
?	Pull filament in before replacing extruder (mm)	0		60	<b>6mm</b>
?	Retraction before move to extruder switch (mm)	0		60	<b>10mm</b>
?	Push the filament out when changing material color (mm)	30		100	<b>55mm</b>
?	Minimum time to print one layer (s)	0		20	<b>10s</b>
?	Infill creation angle (°)	0		90	<b>90°</b>
?	Extrusion width (mm)	0.35		0.45	<b>0.4 mm</b>
?	First layer thickness (mm)	0		0.3	<b>0.15 mm</b>
?	Retraction speed (mm/s)	1		90	<b>40 mm/s</b>
?	Retraction length (mm)	0		40	<b>1.5 mm</b>
?	Distance of supports from model in axes X, Y (mm)	0		5	<b>0.7 mm</b>
?	Distance of supports from model in axis Z (mm)	0		2	<b>0.1 mm</b>

**Shell thickness** - Defines the thickness of the exterior walls of the printed model. The value determines the strength of the wall and therefore overall strength of the model.

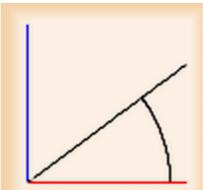
**Top solid layers** - Defines the number of layers on of the top of the model. The strength of the enclosing depends on the quality of the print (layer height).

**Bottom solid layers** - Defines the number of layers on the bottom of the model. The strength of the enclosing depends on the quality of the print (layer height).

**Infill density (%)** - Filling quantity affects the final strength of the model. For greater strength, enter a higher value. Higher values increase printing time.

**Support type** - Specifies the printing of supports. Grid = grid, lines = linear structure. Use grid for tall thin supports. Grid supports are harder to remove.

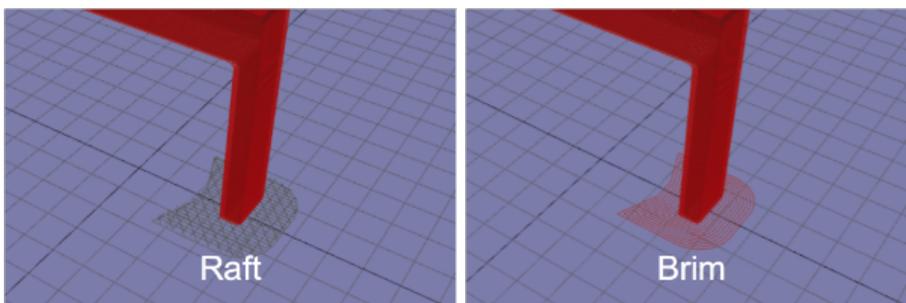
**Angle of support creation (°)** - The supports will only be generated if the angle between the model part and the print bed is smaller than the specified angle.



**Support density** - Support density has an effect on the precision of model part support. A higher density will ensure less spacing in support construction and extend printing time and material consumption. Use higher values for tall thin supports as this should prevent them from falling of mid-print.

**Support everywhere** - Supports are generated only between the model and the bed, or anywhere required by the angle (i.e. even inside the model).

**Initial structure** - Initial platforms help to improve adhesion to the print bed. Brim adds a single layer flat area around the base of your model to stabilise it. Raft adds a thick grid below the object to create a strong foundation for your model.



**Raft density (%)** - Specifies the quantity of material in the initial raft structure.

**First layer air gap** - Selecting the appropriate air gap can simplify removing a raft of model.

**Support printing speed** - Model support printing speed.

**Infill printing speed** - Printing speed of the internal parts of the model.

**Use fan** - Turns printing air cooling on or off.

**Outer perimeter printing speed** - Speed used in printing of external perimeters. Lower printing speed can lead to a finer, more precise, surface.

**Inner perimeter printing speed** - Speed used for printing of internal perimeters. Higher printing speed contributes to shorter printing time.

**Solid layer printing speed** - Print speed of top and bottom layers of the model.

**Temperature of first nozzle** - The temperature of the main nozzle (overrides the predefined setting for a selected material).

**Turn on bed heating** - It helps against peel off of the model (not applicable to Y Soft eDee)

**Bed temperature** - Temperature of bed (not applicable to Y Soft eDee).

**Chamber temperature** - The temperature at which the printer starts to ventilate the chamber.

**Axis Z offset during extruder exchange** - The distance of the print head from the print bed when changing the extruder (not applicable to Y Soft eDee).

**Pull filament in before replacing extruder** - The length of the pulled-in filament when changing extruders (not applicable to Y Soft eDee).

**Retraction before move to extruder switch** - The length of the pushed-out filament when changing extruders (not applicable to Y Soft eDee).

**Push the filament out when changing material color** - Length of pushed-out filament when changing material color (not applicable to Y Soft eDee).

**Minimum time to print one layer** - Increase when printing small objects or details. Useful when the material has to cool down before the printing of the next layer.

**Infill creation angle** - The infill of the model will be generated at the angle specified with regard to the print bed.

**Extrusion width** - Width of extruded material.

**First layer thickness** - Properly chosen thickness may prevent the model from falling off.

**Retraction speed** - Speed of the filament retraction.

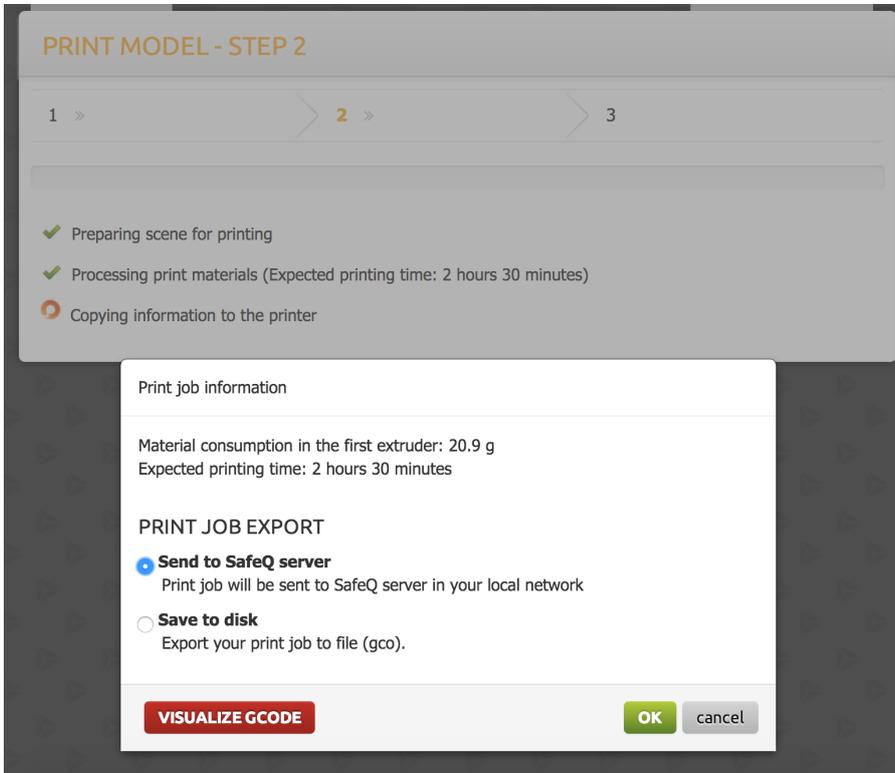
**Retraction length** - Length of the filament retracted before travel.

**Distance of supports from model in axes X,Y** - Distance of the supports from the model. Higher values guarantee easier removal of supports from the finished model, but the surface under the supports will be of a lower quality.

**Distance of supports from model in axis Z** - Distance of the supports from the model. Higher values guarantee easier removal of supports from the finished model, but the surface under the supports will be of a lower quality.

## 8 SAVING OR SENDING THE PRINT JOB TO SAFEQ

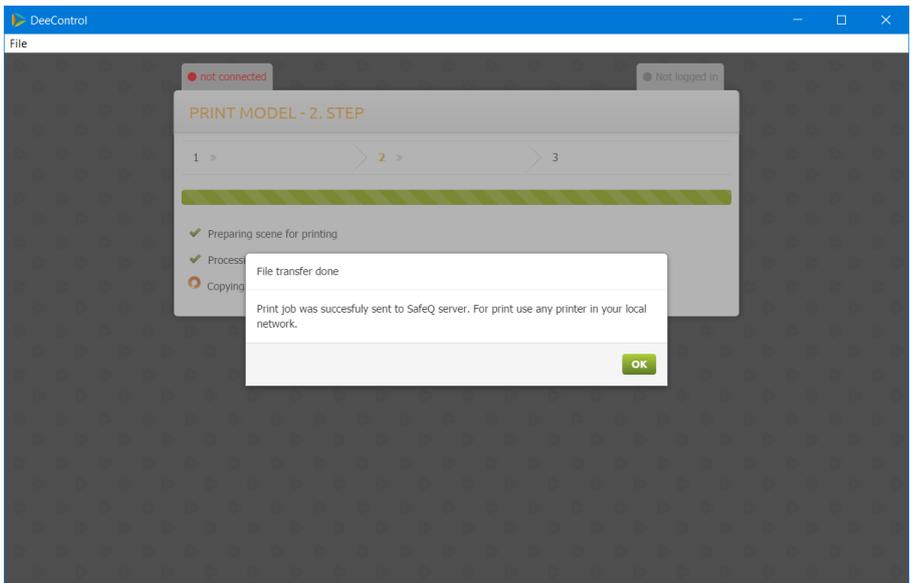
DeeControl prepares the print job and displays information about the estimated amount of filament needed and a time to print.



User can save the job to desired location (e.g. for re-printing) or send it to SafeQ to be printed.

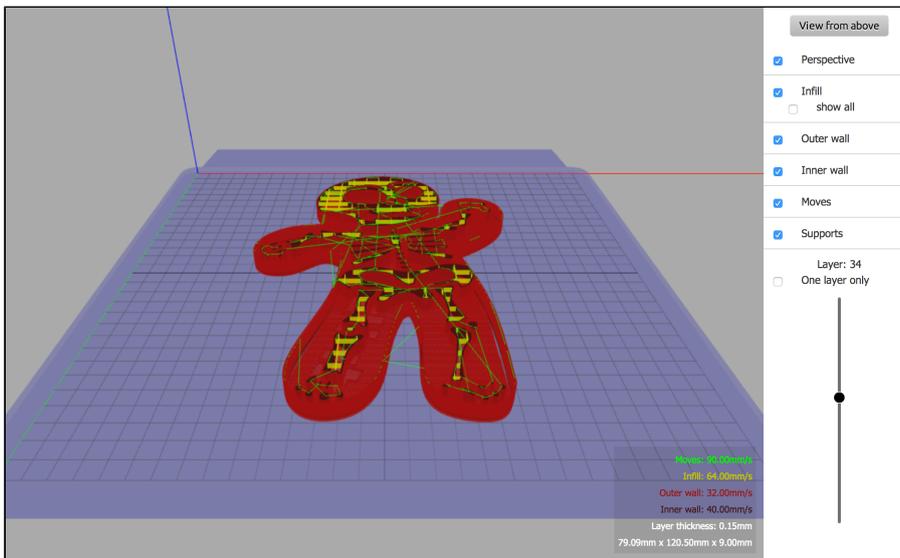
Select the desired option and then **OK**.

In case of sending the job to SafeQ: If the method of authentication is set to domain level accounts, the job will be sent to YSoft SafeQ automatically. Otherwise user have to enter domain credentials into a popup screen of the SafeQ Client, which confirms his identity and the job is added to his account.



## 9 GCODE PREVIEW

Once the print job is processed user can preview/check the generated GCode. This is very useful in case of printing a complex structure with need for support, reviewing the GCode user can check whether the supports have been generated correctly and in all the right places. If user is not satisfied with the outcome, he can always go back to the model edit screen and fine tune the settings or the position of the model.



User can scroll through the whole object layer by layer (moving in the Z axis direction) and select multiple elements to be displayed:

- Infill
- Outer wall
- Inner wall
- Moves (of the print head)
- Supports