3DGence
SLICER 4.0

User manual

Contents

I. GENERAL INFORMATION

1. INTRODUCTION

A dedicated software, **3DGence SLICER 4.0**, was prepared for 3DGence printers. The software is intended to prepare machine codes from files which describe stereometry. It also comprises ready print settings for dedicated materials. Each of the profiles available in 3DGence SLICER 4.0 was developed by a team of specialists and further tested for over 1,000 printing hours of many complicated test models. Therefore, we recommend the use of default profiles when commencing work with 3D printing.

For advanced users, the option of changing the print settings has been provided. The option is intended for modifying the existing profiles such as to adjust the printing quality/parameters to those materials which have not been certified by the manufacturer. Owing to the nature of the parameters modification, the manufacturer does not guarantee the quality or repeatability of the prints generated in that manner.
Please remember that the use of materials and print settings which have not been supplied by the manufacturer takes place at the liability of the user. This means that the manufacturer does not provide support regarding the use of advanced profiles.

3DGence SLICER 4.0 avails of a modified cutting engine – CuraEngine – with the names of parameters maintained, which means that experienced users may avail of an extensive Cura software family documentation. The name and the functions are identical as in the Cura software.

2. QUALITY GUARANTEE

The manufacturer guarantees the possibly highest quality of the models produced with the use of the dedicated software and materials, as well as the prepared default settings. The manufacturer provides also full support in using the print profiles within the software and the recommended printing materials. However, if during the use of the software the user discovers faults in the printed model or software errors, please contact us with the use of the report form available on the www.3dgence.com/support site, with a photo and description of the defect attached and, if possible, a .gcode/.3dg and .stl file (the form is available after an account has been created and the device registered). Each model sent to the manufacturer in that manner will be assessed and/or printed at the company. The manufacturer is going to propose the problem solution by way of an advice, commencement of service activities (if needed), preparing the executable file .gcode/.3dg or print profiles update.

3. INSTALLATION

The software may be downloaded from the manufacturer's website: www.3dgence.com/support. The recommended system requirements to start up the software are:
- Windows 7 or higher, macOS 10.13 or higher,
- screen resolution of 1,920 × 1,080 pixels,
- 4 GB RAM,
- Inter Core i3 double-core processor or higher,
- recommended graphics adapter supporting OpenGL 4.2 or higher.

It is possible to launch the software on hardware which does not fulfil the requirements, however, the comfort of work and the speed of models processing may be deteriorated. The manufacturer does not provide support for equipment which does not fulfil system requirements, particularly with regard to the older versions of operating systems.

II. SHORT INSTRUCTIONS FOR USE

Below presented are short instructions for use of the 3DGence SLICER 4.0 software (Fig. 1). A detailed description of all of the software functions is provided in subsequent chapters.

1. Select a printer for printing the model.
2. Add the model or models for printing.
3. Make sure that the models fit the working field, with the use of automatic models arrangement option.
4. Arrange or modify models on the heatbed. For example, models may be moved, scaled, rotated.
5. Select:
    - module,
    - model material,
    - support material,
    - visual quality,
    - durability.
6. Adjust the print settings if you are at least an intermediate-level user.
7. Prepare your model for printing.
8. Save the model in your computer memory.
9. Download the model to an adequate storage medium.
10. For offline printers, the "e-store" button is available here, which links to the 3DGence's online store. For online printers (ex. 3DGence INDUSTRY F420) the "online printers" button opens the online printing management window.
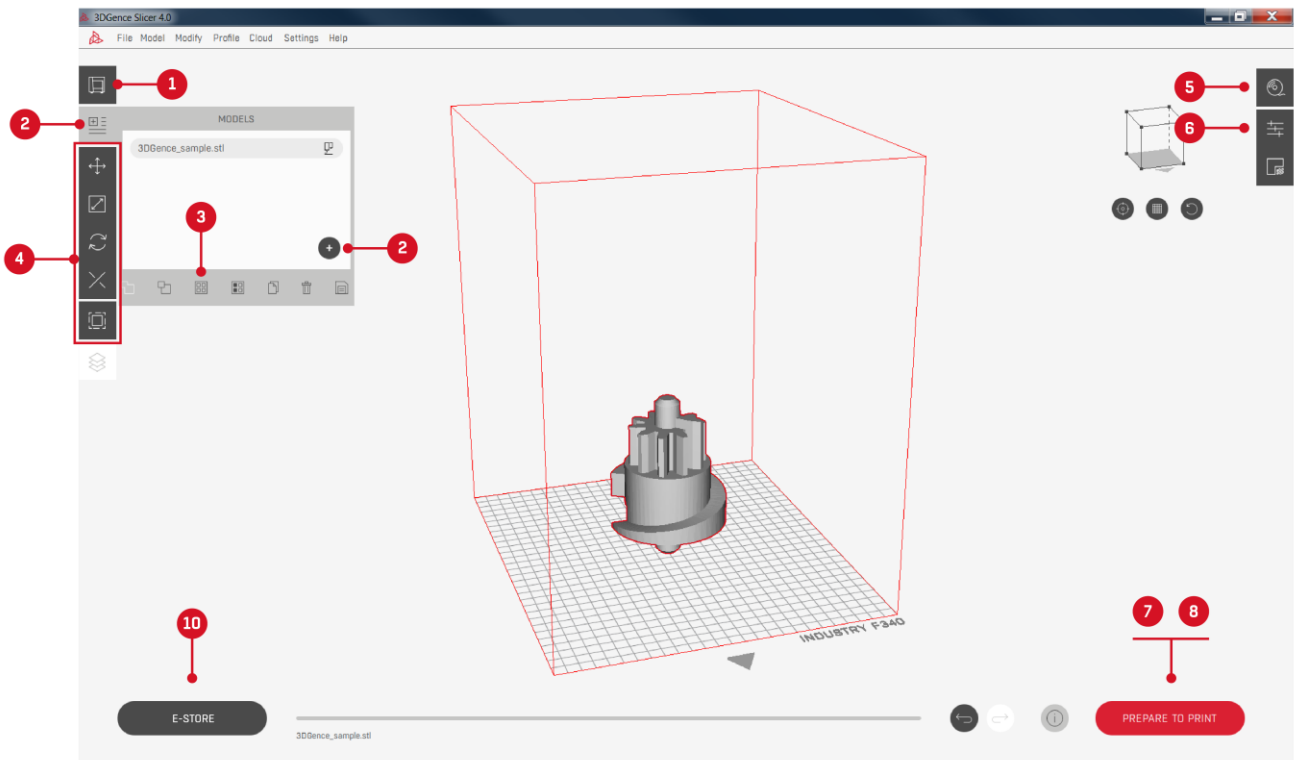

Fig. 1 Software interface

III. SOFTWARE INTERFACE

1. MODEL CONTROL BUTTONS AND KEYBOARD SHORT-CUTS

- The left mouse button selects or deselects a model.
- Pressing the Shift key and selecting a model with the left mouse button enables selection of many models.
- Pressing the right mouse button enables camera rotation.
- The scroll wheel enables the image zooming in or out.
- Pressing of the scroll wheel enables moving of the camera view.
- Buttons which are not active at the moment (e.g. due to the fact that a model has not been selected) are grey.

Short-cut in Windows	Short-cut in macOS	Description
Ctrl + [o]	Cmd + [o]	Open a folder to add a model
Ctrl + [c]	Cmd + [c]	Duplicate the selected model
Ctrl + [z]	Cmd + [z]	Undo
Ctrl + shift + [z]	Cmd + Shift + [z]	Repeat
Ctrl + [p]	Cmd + [p]	Prepare to print
Ctrl + [s]	Cmd + [s]	Save as .stl file
Ctrl + [l]	Cmd + [l]	Activate the function 'Align face to bed'
Ctrl + [e]	Cmd + [e]	Open the advanced options editing window for the selected profile
Ctrl + [r]	Cmd + [r]	Automatic arrangement of models in the middle of the heatbed
Ctrl + [a]	Cmd + [a]	Select all models
Ctrl + up arrow	Cmd + up arrow	Select the previous model in the open list of models
Ctrl + down arrow	Cmd + down arrow	Select the next model in the open list of models
Esc	Esc	Deselection of the selected model
Delete	Backspace	Delete the selected model
Backspace	Backspace	Delete the selected model
Ctrl + [d]	Cmd + [d]	Delete all models
[+]	[+]	Zoom the model in
[-]	[-]	Zoom the model out
Right arrow	Right arrow	Rotate the virtual heatbed right
Left arrow	Left arrow	Rotate the virtual heatbed left
Up arrow	Up arrow	Rotate the virtual heatbed up. In the layers preview mode (after the model has been cut), the next layer may be seen.
Down arrow	Down arrow	Rotate the virtual heatbed left. In the layers preview mode (after the model has been cut), the previous layer may be seen.

2. INTERFACE STRUCTURE

Below presented is the basic structure of the software interface, so as the user may easily locate the specific options (Fig. 2). A detailed description is provided in the subsequent chapters of the instructions.

1. Top bar.
2. Virtual heatbed.
3. An arrow determining the physical front of the printer.
4. Printer selection.
5. Adding models.
6. Loaded model.
7. Models arrangement on the heatbed.
8. Models scaling.
9. Models rotation.
10. Models division.
11. Wall offset.
12. Preview of layers after they have been cut.
13. Selection of model, model and support material, visual quality and durability.
14. Intermediate-level print settings.
15. Support structures.
16. Status bar.
17. Undo button.
18. Repeat button.
19. Information window.
20. Prepare for printing / Save to file.
21. E-store button for printers without online connection or online printers button for models with online connectivity.
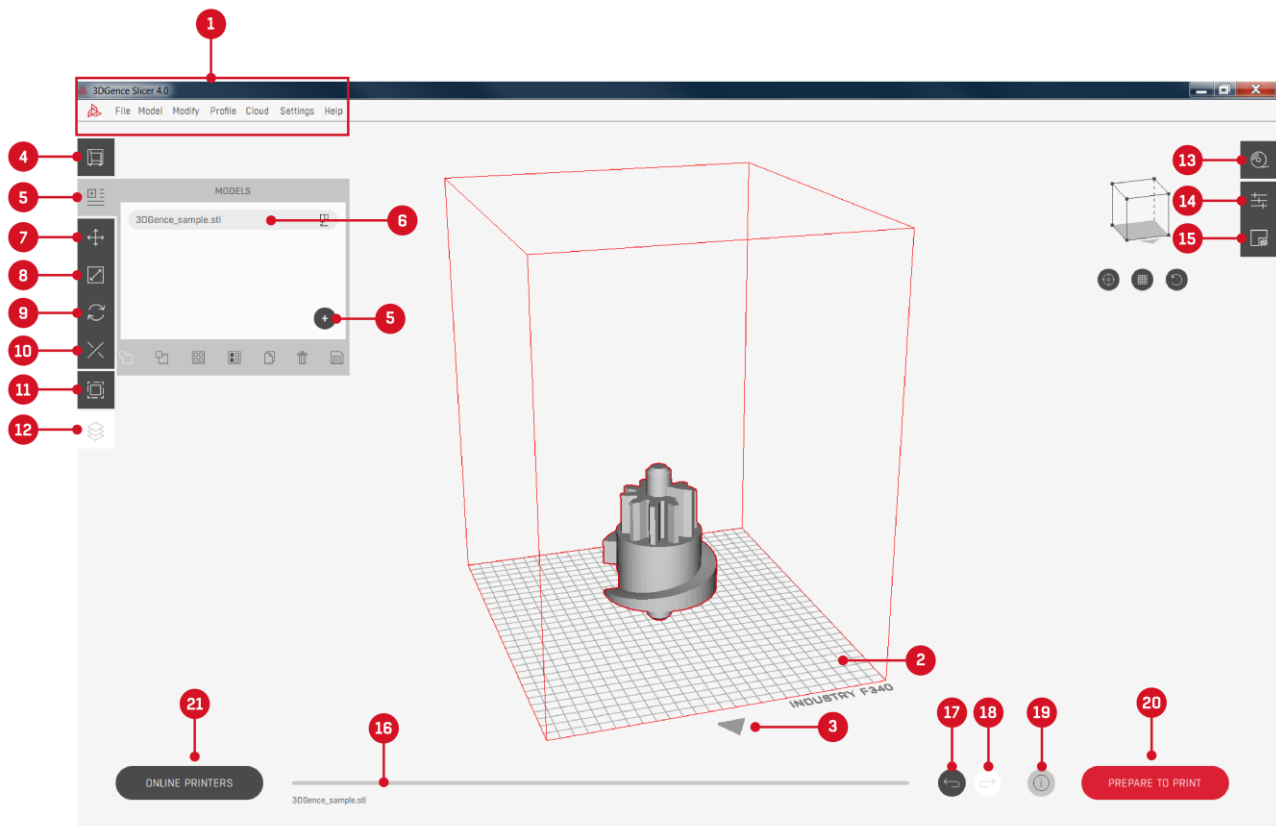

Fig. 2 Software interface

3. LEFT FUNCTIONS BAR

3.1. 'Printers' button

The *'printers'* button (Fig. 3): enables selection of the type of printer for which the model to be printed will be prepared. The printers differ for example with the size of the working field or the prepared print profiles. When a device is changed, the virtual working field will also be automatically changed, and the default material profile loaded for the respective device.
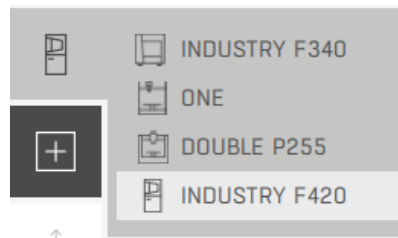

Fig. 3 'Printers' button

3.2. 'Add model' button

The *'Add model'* button (Fig. 4): enables loading files to the software in the following formats: .stl, .obj, .3mf. As soon as at least one model is loaded, the icon on the side bar changes and after selecting the same, a list of the loaded model is displayed. Files may also be added to the software by dragging them to the main software window.

1. A button indicating that the model is active and will be printed. Change of the model from active to an inactive one is possible by selecting the icon with the left mouse button.
2. A button indicating that the model is inactive and will not be printed. Change of the button from inactive to an active one is possible by selecting the icon with the left mouse button.
3. *'Add model'* – enables adding further models to the software.
4. *'Merge models'* – enables merging models in one file. The function is available if at least two models are selected.
5. *'Divide model'* – enables dividing models in .stl files comprising many geometries, such as mergers, into separate files. The file must have at least two closed meshes. The separated files appear on a list of models and from that moment they may be freely modified or saved separately.
6. *'Auto place models'* – it enables automatic arrangement of the loaded models on the printed heatbed, such that they do not overlap. The function leaves the bases of the objects on the heatbed. Models are placed in the left front corner of the working field.
7. *'Multimaterial options'* – enables activation of the options of advanced two-material printing.
8. *'Duplicate model'* – enables duplication of the selected model.
9. *'Delete model'* – enables deletion of the selected model.
10. *'Save mode as .stl'* – enables saving of the model/models as an .stl file.
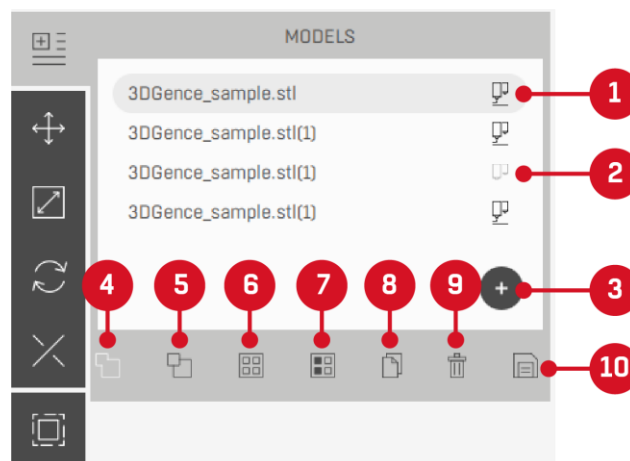

Fig. 4 'Add model' button

3.3. 'Move' button

The *'Move'* button (Fig. 5): enables moving of the selected model in the virtual working area.

The +/- keys enable moving of the selected model by 1 mm in X, Y, and Z axes. It is also possible to move a model any distance by entering the value in the text field. The starting point (0, 0, 0) is located in the middle of the working field.

It is also possible to move models with the use of a mouse – the model must be selected with the left mouse button and with the button pressed it is possible to move the model within XY planes.

It is possible to put models one on top of the other, however, it is not advisable. Moving of any model outside of the working area means that the model will not be printed. The model which does not fit the heatbed will be highlighted red.

1. *'Align face to bed'* – enables identification of the wall which will become the base of the printing.
2. *'Place'* – puts the model on the heatbed.
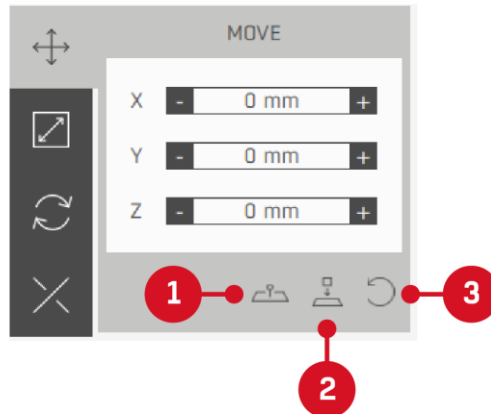3. *'Reset'* – return to the default settings.

Fig. 5 'Move' button

3.4. 'Scale' button

The *'Scale'* button (Fig. 6): enables changing the size of the selected model.

The +/- keys enable scaling of the selected model by 1 mm or 1% in X, Y, and Z axes. It is also possible to scale a model by any value by entering the value in the text field.

1. *'Uniform on'* – enables to scale the model in all axes by the same value. The *'uniform off'* button activates the option and it is possible to scale only along one axis. Change of the option is possible by selecting the icon with the left mouse button.
2. *'Reset'* – return to the initial values of the scaled model, identical to those at the moment of loading the .stl file.
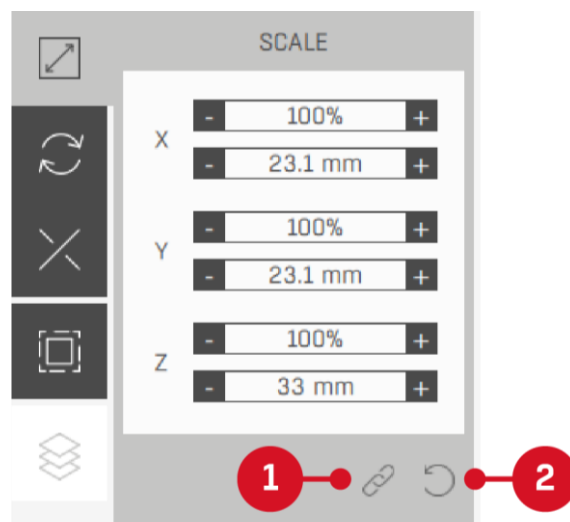

Fig. 6 'Scale' button

3.5. 'Rotate' button

The *'Rotate'* button (Fig. 7): enables rotation of the selected model.

The +/- keys enable rotation of the selected model by 45° in X, Y, and Z axes. It is also possible to rotate a model by any angle by entering the value in the text field.

1. *'Align face to bed'* – enables identification of the wall which will become the base of the printing.
2. *'Place'* – puts the model on the heatbed.
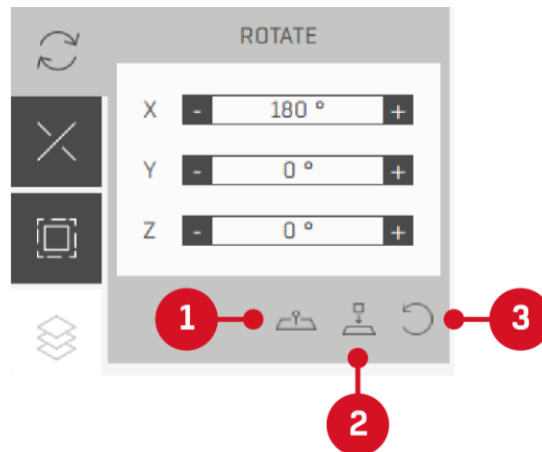3. *'Reset'* – return to the default settings.

Fig. 7 'Rotate' button

3.6. 'Split' button

The *'Split'* button: enables dividing the selected model into parts. The use of the option is recommended particularly in a situation when the model does not fit the working area of the device.

3.6.1. Simple split

The *'Simple split'* option (Fig. 8, 9): divides a single file into parts along the selected cutting plane (X, Y or Z).

The +/- buttons and the slider enable moving of the cutting plane. As soon as the plane is set, the option *'Split model'* must be selected. If the file may not be properly divided along a plane, the status bar will show a respective message – in such case the user must make sure that the .stl file has no errors. After division, the model will be highlighted yellow. This means that two new models are too close to each other and must be separated manually or with the use of the *'Auto place models'* option.
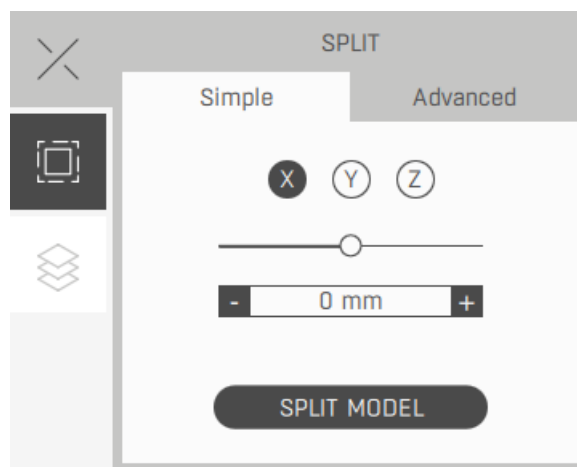

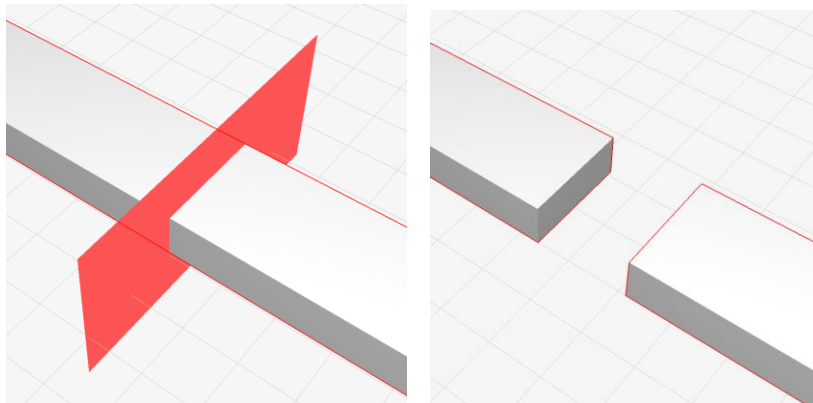*Fig. 8 'Split' button – Simple split*

Fig. 9 Simple split

3.6.2. Advanced split

The *'Advanced split'* option (Fig. 10, 11): divides a single file into parts along the selected plane and creates notches on both elements, so that they may be easily joined after printing, for example by gluing.

The +/- buttons and the slider enable moving of the cutting plane. As soon as the plane is set, the option *'Split model'* must be selected. If the file may not be properly cut along a plane, the status bar will show a respective message – in such case the user must make sure that the .stl file has no errors.

Owing to the differences in sizes and shapes of the models, the division parameters must be selected by the user.
1. 'Edge' – the distance between the notch and the model edge (A).
2. 'Angle' – notch angle (B).
3. 'Offset' – additional distance between the divided model parts, for example to make a joint. It must be remembered that owing to engineering reasons the edges of the print in the place of joining may be rounded, therefore, it is recommended that at least 0.2 mm offset is left (C).
4. 'Depth' – notch depth (D).
5. 'Invert' – change of the direction of the notch.

A model division may be time consuming and will be made if at least one notch may be generated. In case a model cannot be divided, the cutting settings must be changed.
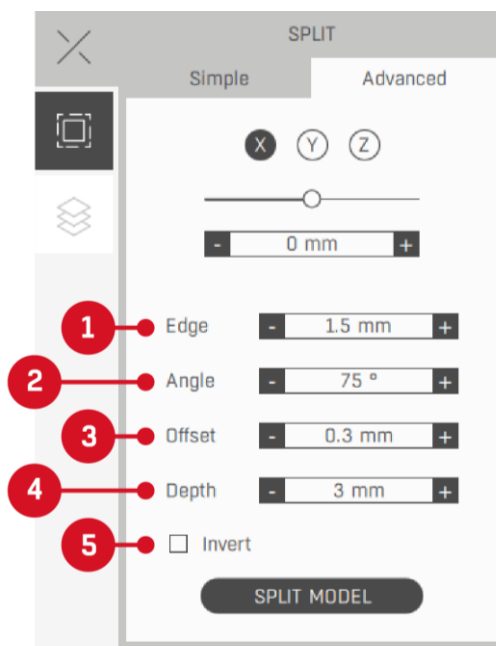
Fig. 10 'Split' button – Advanced split
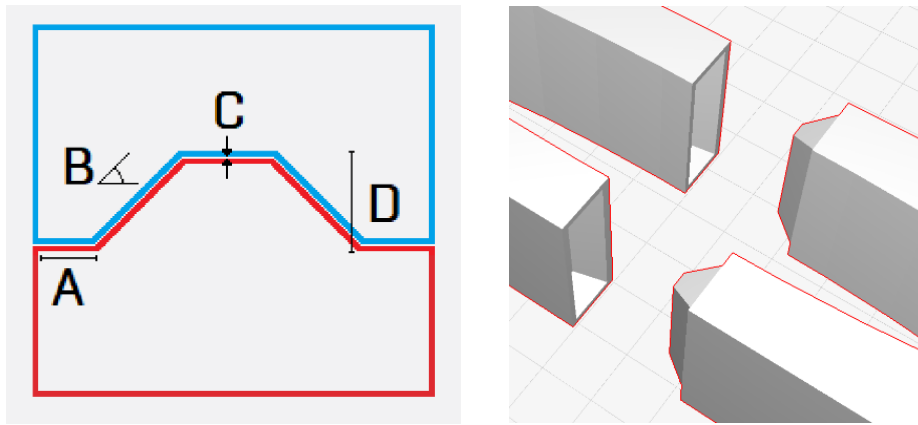
Fig. 11 Advanced split

3.7. 'Wall offset' button

The *'Wall offset'* option (Fig. 12, 13): changes the size of the model by extending or reducing its walls independently of the model size, after a .gcode or .3dg file has been generated. Fig. 13 presents a model loaded to the software, with 3 mm offset applied. The option is recommended to improve the dimensional tolerance of tight-fitting parts.

Fig. 12 'Wall offset' button

Fig. 13 Wall offset

3.8. 'Layers' button

The *'Layers'* option (Fig. 14): enables preview of the layers of the model prepared to print. Before cutting the model the button is inactive (grey).

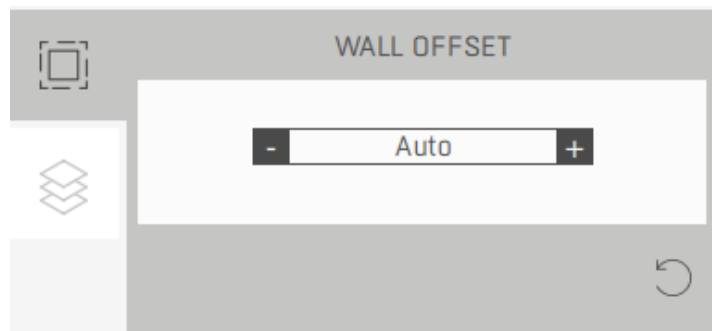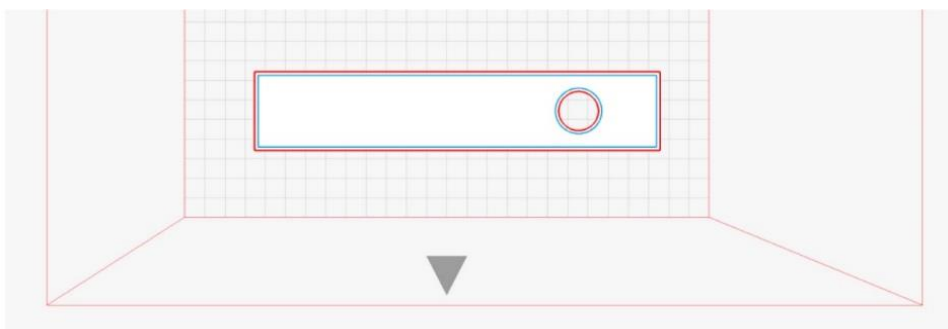The *'Add pause point'* button enables determination after which layer the print will stop. It is also possible to remove the pause point added by the user. The "add pause point" option availability depends on device model.
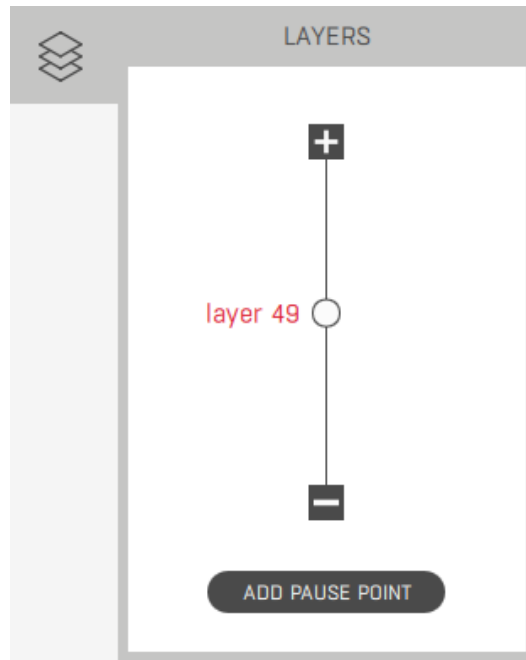

Fig. 14 'Layers' button

4. RIGHT FUNCTIONS BAR

4.1. 'Profile' button

The *'Profile'* button (Fig. 15): enables the selection of:

1. The type of the printing module compliant with the module installed in the printer (e.g. PRO module or 0.4 hotend).
2. The type of the model material, e.g. the material from which the model will be printed.
3. The type of the support material, e.g. the material from which the model will be printed. Single-hotend printers will always print supports from the same material as the target model.
4. The visual quality of the model. Improvement of the visual quality of the model will reduce the height of the printed layer and will improve the visual parameters of the printed model, however, the printing time will be significantly longer;
5. Model durability. Durbility improvement means that the prints will be created from a greater quantity of material and will be stronger. They will be more susceptible to material shrinkage deformation and their printing time will be longer. The type and number of the available settings depends on the selected visual quality.
6. Type of work plate. Different build plate sheets option avability depends on device model.
7. The *'Load custom profile'* button, which enables loading of a material profile from outside of the default (recommended) settings list.
8. The *'Adjust print settings'* button, which takes the user to the tab comprising intermediate-level settings where print settings may be adjusted.
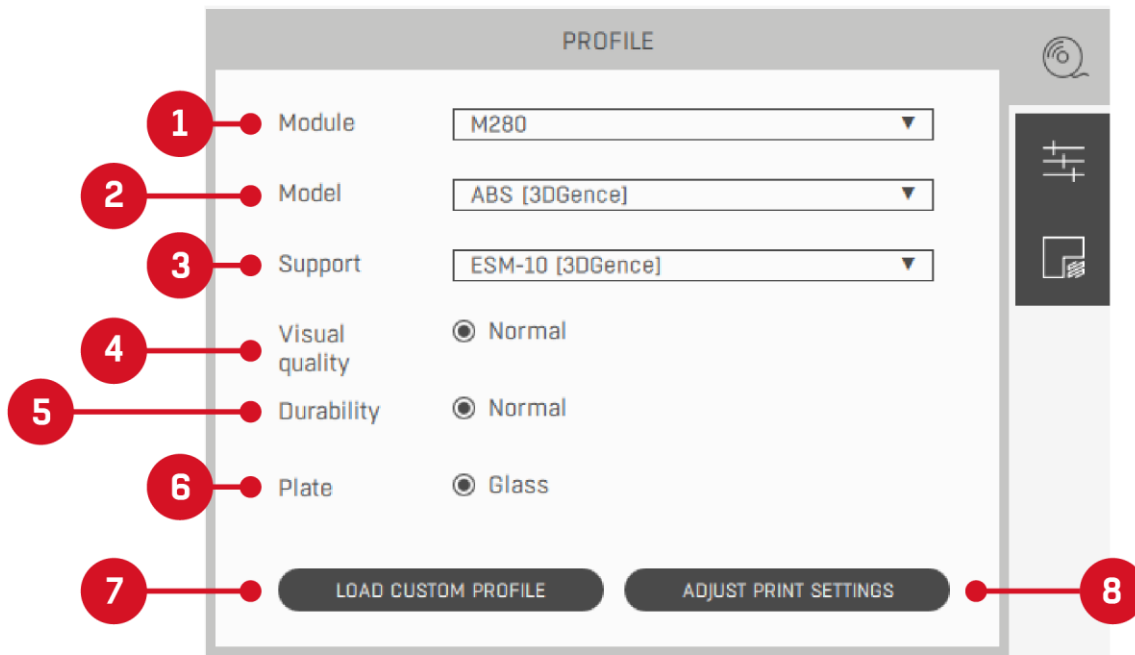
FIg. 15 'Profile' button

4.2. 'Adjust print settings' button

The *'Adjust print settings'* button (Fig. 16): enables the users possessing intermediate-level skills to change the print settings. The option is intended for modifying the existing profiles such as to adjust the settings to the user own needs. It must be remembered, however, that a change of the parameters prepared by the manufacturer may affect the print quality as well as dimensional accuracy.

1. *'Infill percent'* – is the density of print filling specified in percent.
2. *'Infill pattern'* – is the shape of the print filling.
3. *'Side walls'* – is the number of the print side walls.
4. *'Top walls'* – is the number of fully filled top layers of the print.
5. *'Bottom walls'* – is the number of fully filled bottom layers of the print.
6. *'Bed adhesion'* – is the type of the print adhesion to the heatbed. Improper adhesion may result in the print damage and its detachment from the heatbed.
7. *'Info'* – displays a window with a description of the particular parameters.
8. *'Reset'* – reinstates the default settings prepared by the manufacturer.
9. *'Save profile'* – enables saving the user profile with changed parameters on the disc.
10. *'Expert settings'* – enables loading and modification of the material profile from outside of the list of default (recommended) settings. It is an option intended for advanced users. It enables modification of the existing profiles such as to improve quality for materials which have not been certified by the manufacturer or adjust the device working settings. Owing to the nature of the parameters modification, the manufacturer does not guarantee the quality and repeatability of the prints generated in that manner. Please remember that the use of materials and print settings which have not been supplied by the manufacturer takes place at the liability of the user. This means that the manufacturer does not provide support regarding the use of advanced profiles.
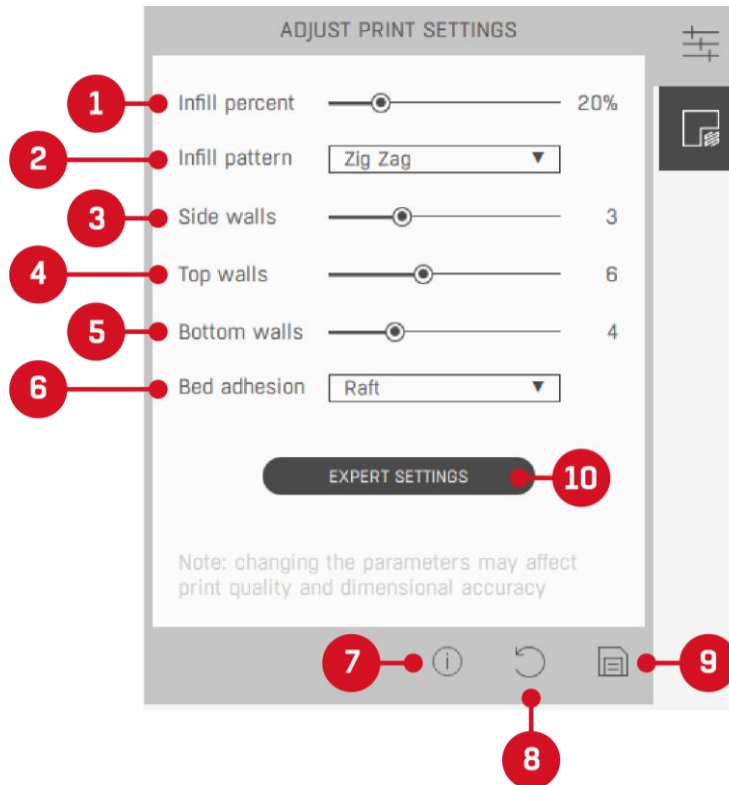
Fig. 16 'Adjust print settings' button

4.3. 'Supports' button

The *'Supports'* option (Fig. 17): enables changes in the support structures. When the window is selected, the *'Auto'* option is visible, which refers to the parameters selected by the manufacturer. After switching into *'Manual'*, it is possible to manually set the angle at which supports will be generated in reference to the heatbed plane. If the model is supported on a very small area of the heatbed, there is a hazard of its breaking, therefore, it is recommended to reduce the parameter in such case. More supports means longer printing time and a more stable print, less supports means shorter printing time and less stable print. A comparison of various support angle values is presented in Fig. 18. In the *'Manual'* option, the *'Rigid support enabled'* option is also available, which generates strengthened supports that are more difficult to remove. The option is recommended in printing models with high supports. In the *'Supports'* window, there is a possibility to completely switch off the supports by selecting the *'Disabled'* option.
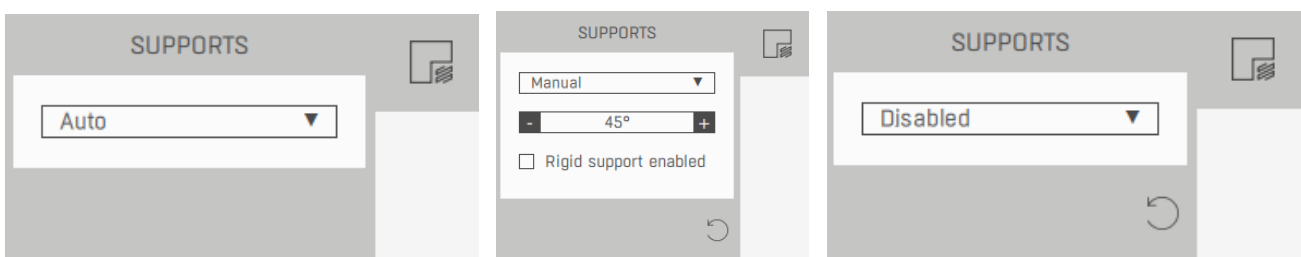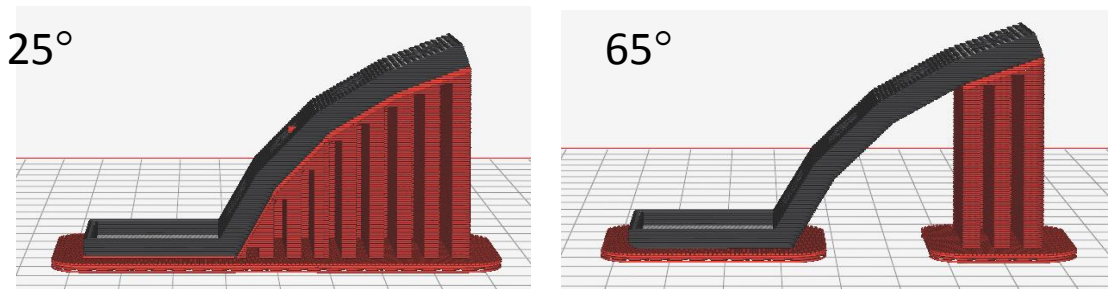

Fig. 17 'Supports' button

Fig. 18 Comparison of the angle of support

5. TOP BAR

Software navigation is also possible be selecting the particular options from the top bar (Fig. 19). Additionally, the bar enables:

- changing of the software language;
- changing of the information window settings;
- checking for updates;
- switching on automatic updates and new versions information;
- redirection to the 3DGence technical support page;
- switching on of the first-use assistant;
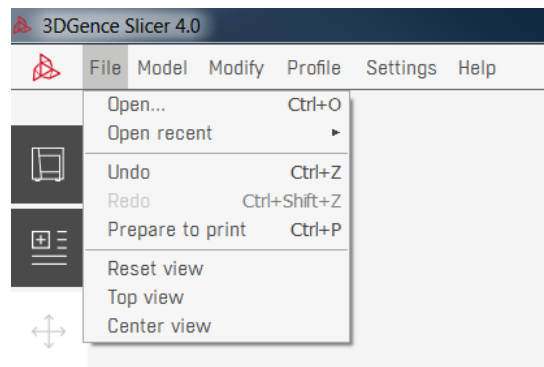- redirection to the software user's instructions download page.

Fig. 19 Top bar

6. BOTTOM BAR

1. Status bar (Fig. 20) – displays the software operation statuses.
2. The *'Undo'* – button reverses the latest operation. If no operations were made with regard to the solid body after the model had been loaded, the selection of the button will result in deletion of the latest loaded .stl model.
3. The *'Redo'* button – reinstates the undone operation.
4. The *'Information'* button – enables preview of the name and dimensions of the model, if the model is selected. If the model is deselected, it enables a preview of the key information about the selected material profile.
5. The *'Prepare to print'* button – generates a .gcode or .3dg executable file for the selected printer and the selected print settings. The status bar will display the process progress bar. After the work completion the image of the solid body in the main window changes because the software switches to the mode of previewing the paths followed by the printer during operation. After preparing the model for printing, the *'Prepare to print'* button will be replaced with a new button: *'Save to file'*, which will enable saving the prepared file in the computer.
6. For offline printers, the "e-store" button is available here, which links to the 3DGence's online store. For online printers (ex. 3DGence INDUSTRY F420) the "online printers" button opens the online printing management window.
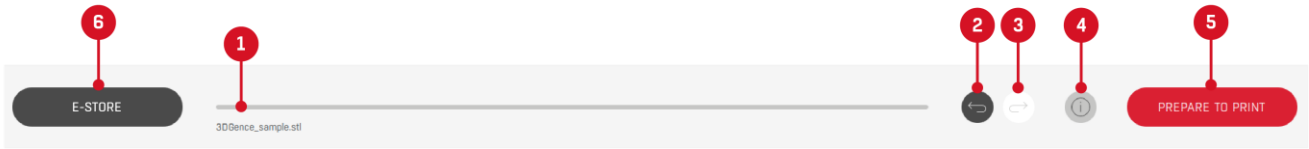
Fig. 20 Button bar

7. VIRTUAL HEATBED

1. The virtual heatbed (Fig. 21) – is a graphic representation of the machine working area, where the models to be printed are set. The software will not allow printing of models which partially protrude outside of the working area. The models which do not fit the heatbed are highlighted red. The size of the area changes dynamically when the printer is selected. Part of the working area is in most cases reserved to the print, e.g. of a raft which improves the print adhesion to the heatbed. It is recommended to print within the distance of at least 5 mm from the working field edge.

2. The arrow on the heatbed specifies the place where the physical front of the printer is located as well as the operator panel.

3. Graphic representation of the model loaded to the software. Overlapping models will be printed, but errors may appear on them. Models which do not fit the working area will not be printed, they must be divided into smaller elements.

4. The cube helps selecting the basic view points.

5. The *'Center* button – focuses the camera on the selected model.

6. The *'Top view'* button – enables viewing the model from the top.

7. The *'Reset'* button – reinstates the camera default settings. It is useful, when during the camera moving with the use of the mouse, the working area gets out of sight.
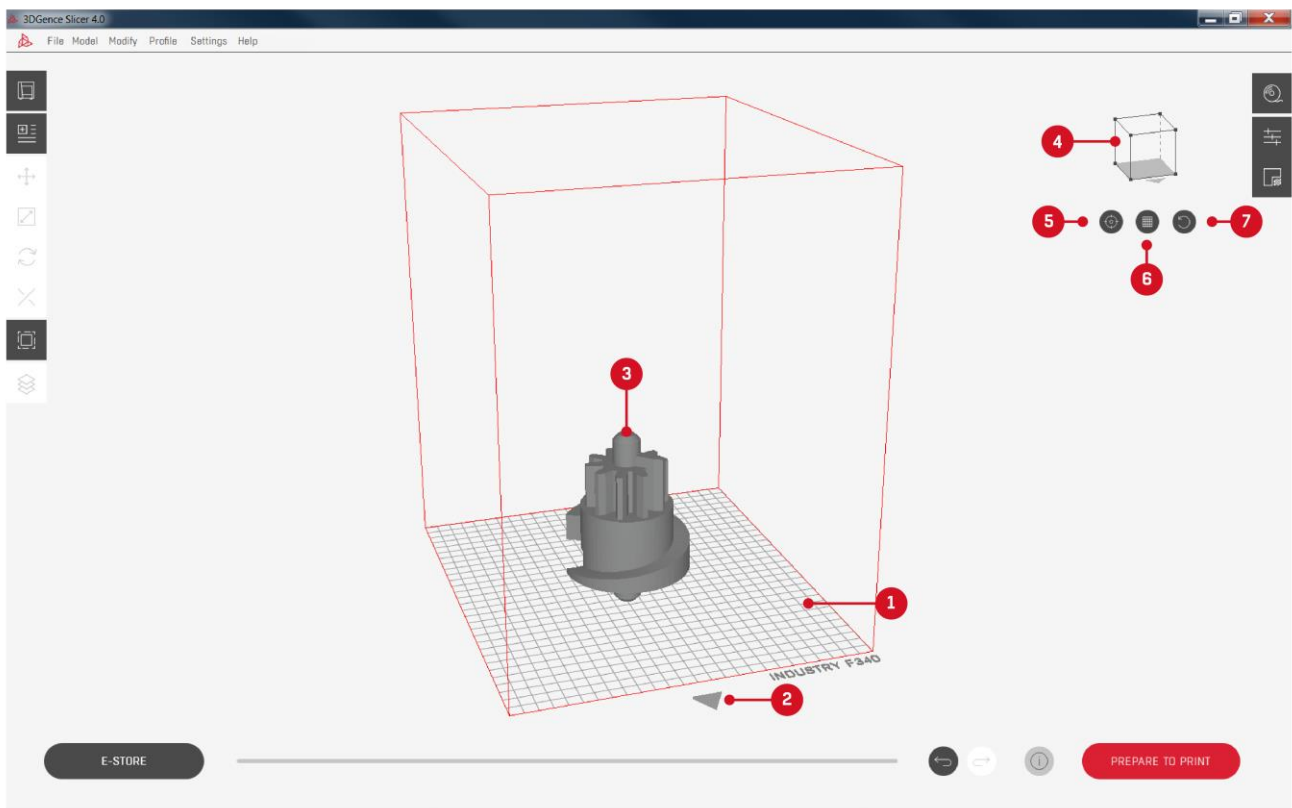

Fig. 21 Virtual heatbed

8. ONLINE PRINTERS

The "online printers" option (Fig. 22) enables sending files to a printer via local network or 3DGence CLOUD system from the 3DGence SLICER 4.0. In fig. 22 it is possible to select "LAN" or "3DGence CLOUD". This selection will be saved until you change the connection.
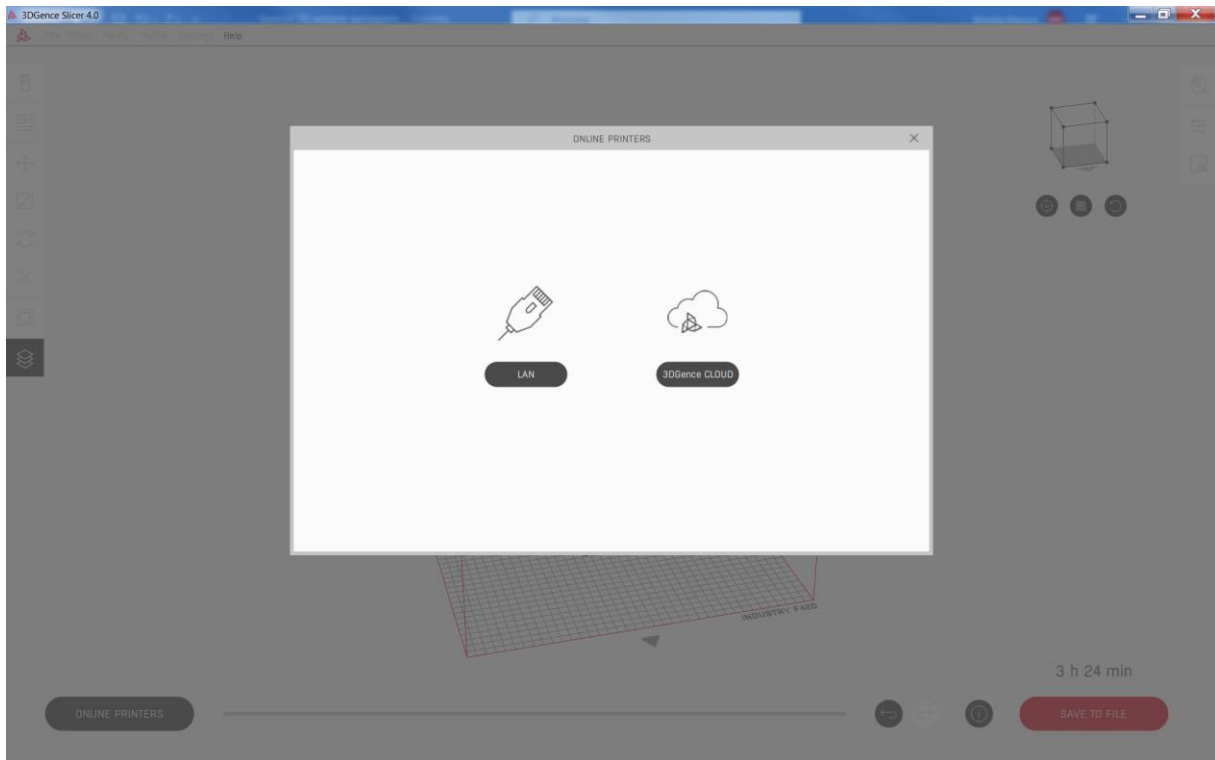
Fig. 22 Online printers option

8.1. Local network

After selecting the LAN option in Fig. 22, the user is redirected to the screen associated with the local network (Fig. 23). The computer and printer should be connected to the same IP domain, so that you can search for printers and send files. At the first start, the list of added printers on the left is empty.

1. The icon marked red indicates whether the user is connected via LAN or 3DGence CLOUD. After selecting the gray icon, you can change the network connection.
2. You can search for a printer by entering its IP address.
3. List of available printers in the network that have been searched.
4. After selecting the printer, the "add printer" button allows you to add the device to the printer list available on the left. If neither printer is selected, the "add printer" button is inactive.

Fig. 23 Screen after selecting LAN - printer search

After adding printers to the list on the left and selecting one of them, related information are on the right (Fig. 24).

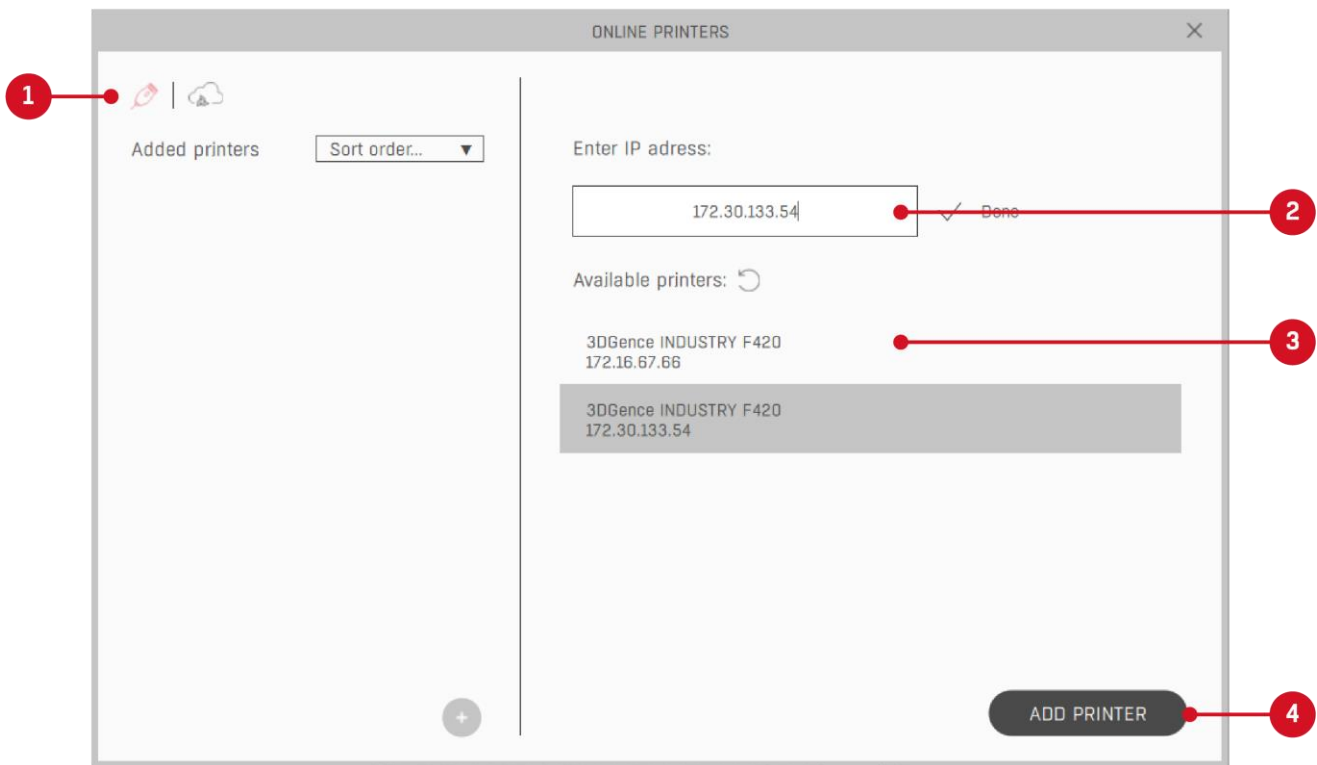1. The icon marked red indicates whether the user is connected via LAN or 3DGence CLOUD. After selecting the gray icon, you can change the network connection.
2. Sort printers by name or state.
3. Added printers.
4. Selected printer.
5. Remove a printer from the list is preceded by an appropriate message.
6. Add a new printer - moves to the screen shown in fig. 23.
7. Camera preview and refresh button.
8. Printer name.
9. Printer serial number.
10. Printer status.
11. Progress of the printing process.
12. Number of prints in the queue, estimated queue time and materials.
13. Files in the print queue.
14. Expanding the queue allows you to see: preview the file. 3dg, file name, printing time, profile on which the file was prepared, and quality and durability.
15. Add file from disk to queue.
16. Adding the file prepared in the software to the queue without saving to disk.
17. Delete file from the queue. Files added by other users cannot be deleted.
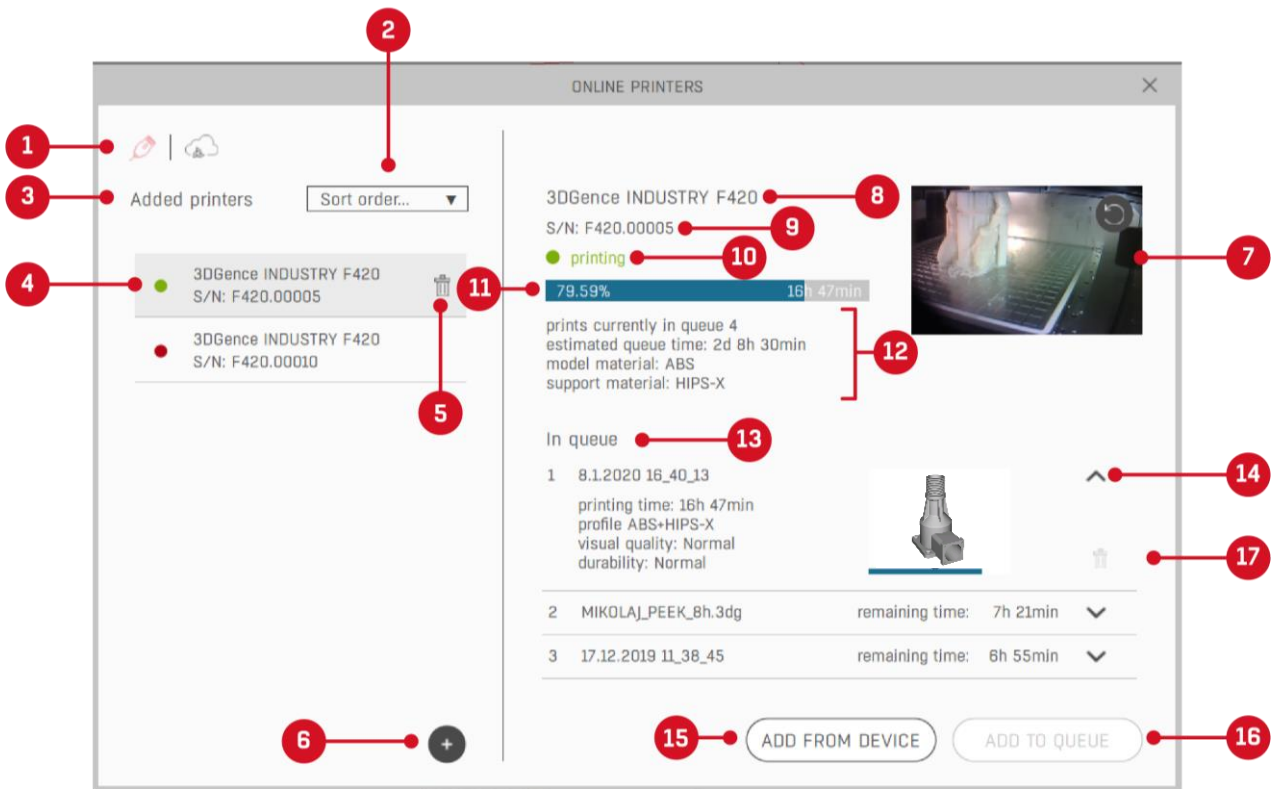
Fig. 24 Screen after selecting LAN option

8.1.1. Transfer the print to the printer from 3DGence SLICER 4.0 through the local network

The prerequisite is to connect the printer and the computer to the same IP domain. No Internet access is required to transfer the print to the printer from 3DGence SLICER 4.0 via the local network.

1. Turn on the printer.
2. Using 3DGence SLICER 4.0 software prepare a model for printing.
3. Select the "online printers" button in the bottom left-hand corner of the software.
4. In the "online printers" window, select "LAN".
5. Add a printer by selecting it from the list of available printers or by entering its IP address.
6. On the left select the printer to which you want to send the print job. The printer must be online.
7. Select "add to queue" to add a print in the software to the queue or "add from device" to add another print that has been saved to the drive.
8. Access the printer.
9. Mount a work surface corresponding to the one selected in 3DGence SLICER 4.0.
10. From the main menu, select the "files" icon or press the centre of the circle in the middle of the display. Both options will access the print queue.
11. Press the file name to select the model you want to print and press "PREVIEW".
12. The model preview will be displayed on the screen where you can press "PRINT" to start printing.
13. The device will automatically start the warm-up procedure and the working chamber door will be locked.
14. When the set temperatures are reached in accordance with the prepared .3dg file and the heating process is completed, the printing process will start automatically.
15. Once the printing process is complete, the printer will start cooling the work chamber, hotends and heated print bed. Wait until this process is completed. Do not remove the print before the machine is fully cooled as it may deform the printed model.
16. Once the cooling process is complete, the working chamber door will be unlocked, and the print can be removed. The print should be removed with the spatula, gently prizing its base.

⚠️ **Note**: The print should be removed when the device cools down to avoid deformation of the printed model and damage to the glass printing surface.

8.2. 3DGence CLOUD

After selecting the 3DGence CLOUD option in Fig. 22, the user is redirected to the screen where he can log to his account in the 3DGence CLOUD system (Fig. 25).
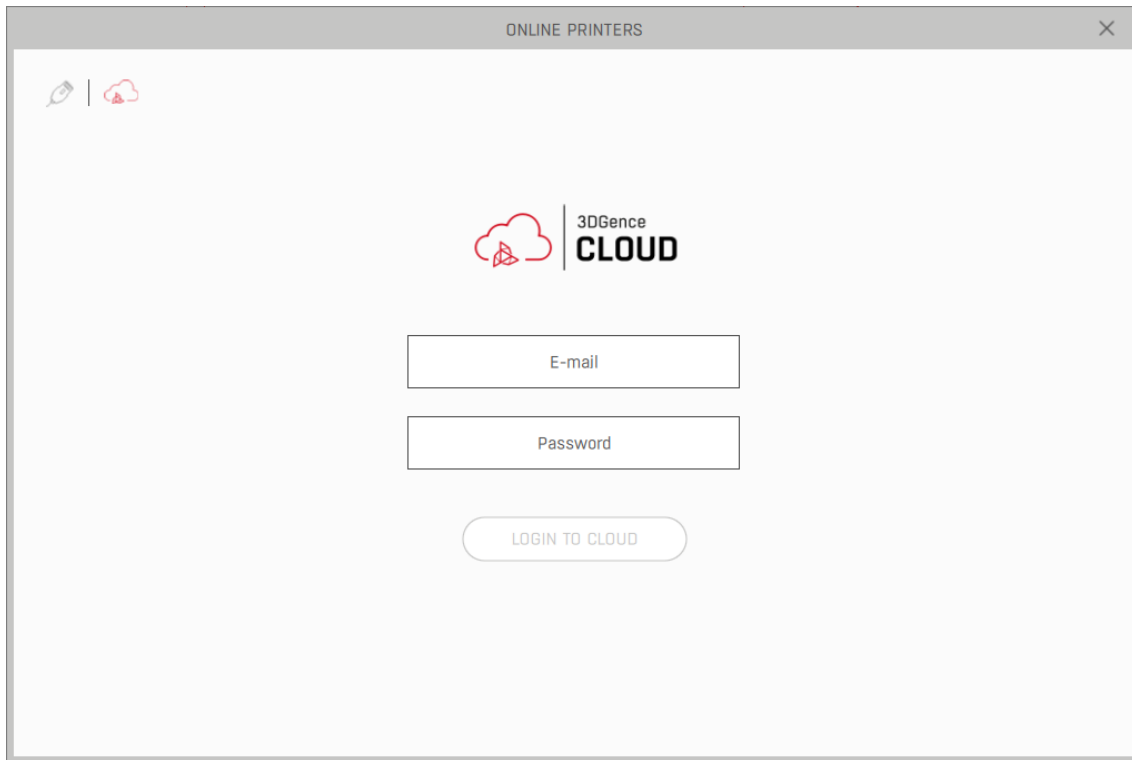

Fig. 25 3DGence CLOUD login screen

After logging in, if printers were added to the 3DGence CLOUD internet platform - list is visible on the left. After selecting the printer, information related to it is displayed on the right (Fig. 26).

1. The icon marked red indicates whether the user is connected via LAN or 3DGence CLOUD. After selecting the gray icon, you can change the network connection.
2. Sort the printers by name or state.
3. Added printers to the 3DGence CLOUD internet platform.
4. Selected printer.
5. Edit the printer, which opens the web platform www.cloud.3dgence.com. From the level of the Internet platform you can, for example, change the name of the printer or delete it.
6. Username.
7. Log out button.
8. Camera preview and refresh button.
9. Printer name.
10. Printer serial number.
11. Printer status.
12. Progress of the printing process.
13. Number of prints in the queue, estimated queue time and materials.
14. Files in the print queue.
15. Expanding the queue allows you to see: preview the file. 3dg, file name, printing time, profile on which the file was prepared, and quality and durability.
16. Add file from disk to queue.
17. Adding the file prepared in the software to the queue without saving to disk.
18. Delete file from the queue. Files added by other users cannot be deleted.
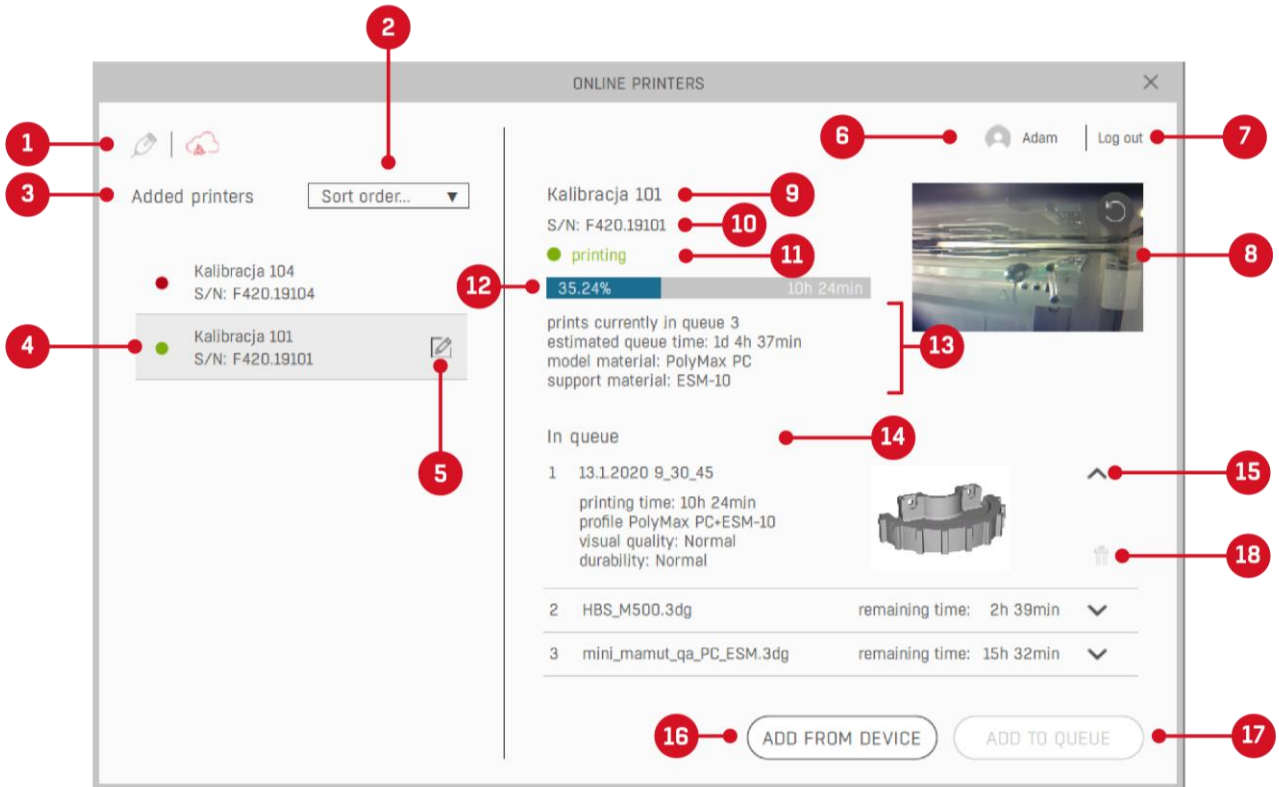
Fig. 26 Screen after logging in to 3DGence CLOUD, when printers are added

If there are no printers added to the 3DGence CLOUD online platform in a given organization, the screen shown in Fig. 27 is displayed. After selecting the "go to cloud" button, the user is redirected to www.cloud.3dgence.com, where he can add a device, which will then be visible in 3DGence SLICER 4.0.
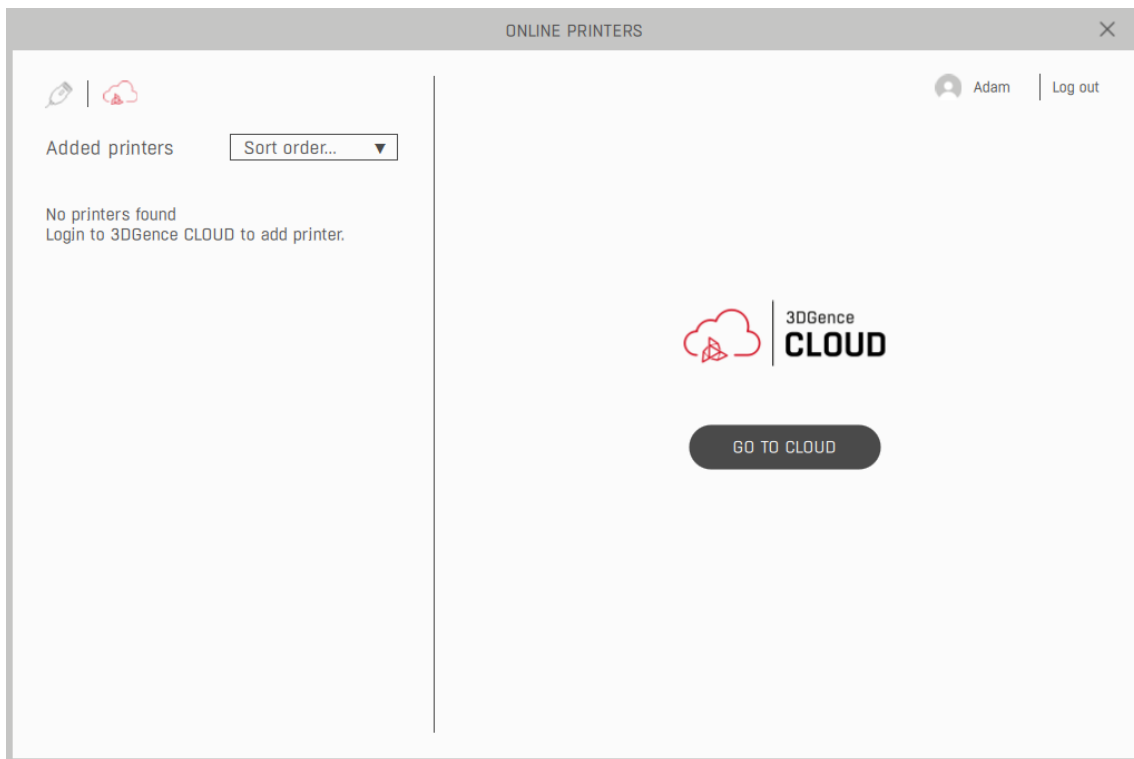

Fig. 27 Screen after logging in to 3DGence CLOUD when printers are not added

8.2.1. Transfer the print to the printer from 3DGence SLICER 4.0 through the 3DGence CLOUD

Sending a print to the printer from 3DGence SLICER 4.0 via CLOUD does not require the printer and computer to be in the same IP domain. A high level of security is ensured as the file is sent in fragments and files are not stored anywhere after being sent by the CLOUD system.

1. Turn on the printer.
2. Using 3DGence SLICER 4.0 software prepare a model for printing.
3. Select the "online printers" button in the bottom left-hand corner of the software.
4. In the "online printers" window, select "3DGence CLOUD".
5. Log in to your 3DGence CLOUD account.
6. On the left select the printer to which you want to send the print job. The printer must be online. If the printer is not added, use the "go to cloud" button to access the 3DGence CLOUD web platform where you can add your device.
7. Select "add to queue" to add a print in the software to the queue or "add from device" to add another print that has been saved to the drive.
8. Access the printer.
9. Mount a work surface corresponding to the one selected in 3DGence SLICER 4.0.
10. From the main menu, select the "files" icon or press the centre of the circle in the middle of the display. Both options will access the print queue.
11. Press the file name to select the model you want to print and press "PREVIEW".
12. The model preview will be displayed on the screen where you can press "PRINT" to start printing.
13. The device will automatically start the warm-up procedure and the working chamber door will be locked.
14. When the set temperatures are reached in accordance with the prepared .3dg file and the heating process is completed, the printing process will start automatically.
15. Once the printing process is complete, the printer will start cooling the work chamber, hotends and heated print bed. Wait until this process is completed. Do not remove the print before the machine is fully cooled as it may deform the printed model.
16. Once the cooling process is complete, the working chamber door will be unlocked, and the print can be removed. The print should be removed with the spatula, gently prizing its base.

⚠️ **Note**: The print should be removed when the device cools down to avoid deformation of the printed model and damage to the glass printing surface.

IV. EXECUTABLE FILE PREVIEW MODE

The view after the model has been prepared for printing enables the print preview, which means the shape in which the paths during 3D printing will be followed (Fig. 28). The slider enables viewing of the layers generated by the software. In the preview, it may be checked whether the model will print properly. For example, if the whole large model is supported only on a few square millimetres of support, it will probably collapse when printing, in such case the user must go back to the main interface and enlarge the generated supports.

1. Preview of the model layers.
2. Representation of the expected paths of the material from a .gcode or .3dg file. The slider enables changing of the preview of the active model layer.
3. A button enabling adding or deleting a pause in print.
4. Getting back to the main interface before cutting a model. After getting back, the model and the settings may be edited, but another executable file must be prepared before the process continues.
5. The expected printing time.
6. Information window.
7. A button for saving the model to a file. As soon as the button is selected, a window will appear where the inventory and the name of the file may be selected.
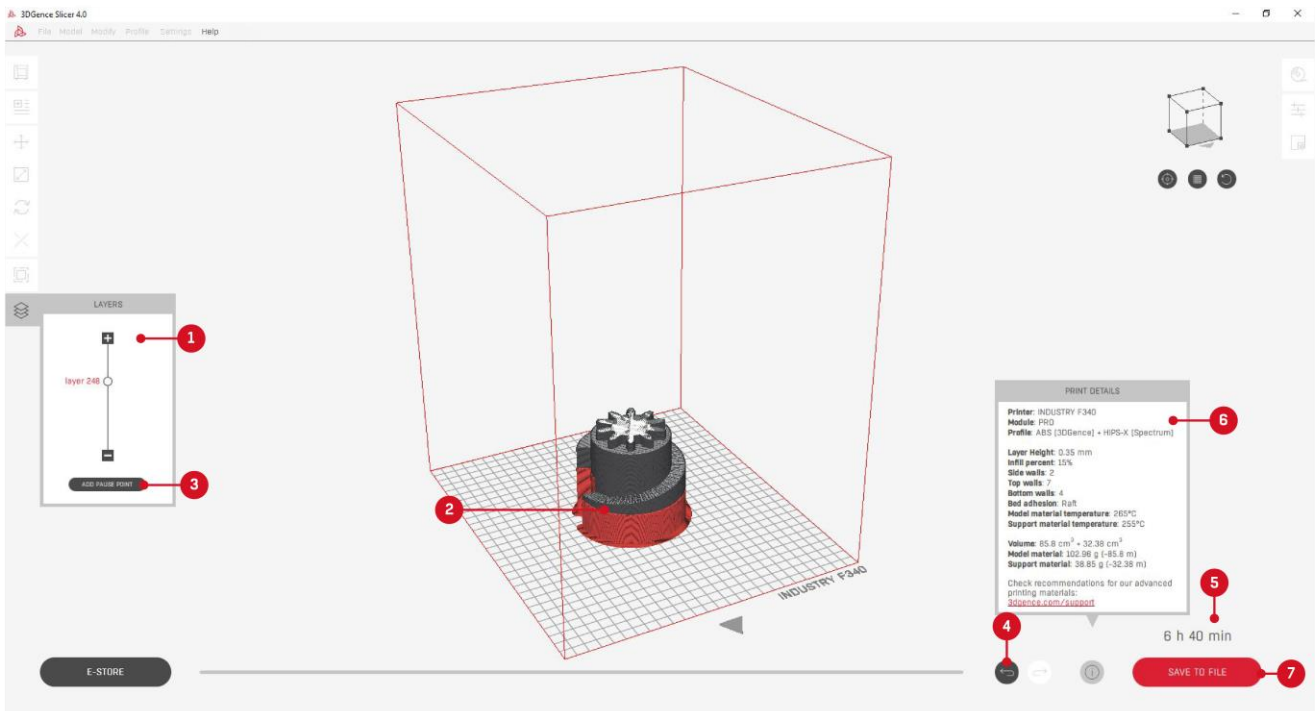

Fig. 28 Executable file preview mode

V. ADVANCED SETTINGS

For advanced users, the option of changing the print settings has been provided. It enables modification of the existing profiles such as to improve quality for materials which have not been certified by the manufacturer or adjust the device working settings. Owing to the nature of the parameters modification, the manufacturer does not guarantee the quality and repeatability of the prints generated in that manner.

Please remember that the use of materials and print settings which have not been supplied by the manufacturer takes place at the liability of the user. This means that the manufacturer does not provide support regarding the use of advanced profiles.

3DGence SLICER avails of a modified CuraEngine with the names of parameters maintained, which means that experienced users may avail of an extensive Cura software family documentation. The name and the functions are identical as in the Cura software.

1. LOADING OF ADVANCED PROFILES

1. Select the *'Load custom profile'* option in the *'Profile'* window or select the option from the top bar.
2. Read and confirm the warning message.
3. From the list, select the profile you want to edit (Fig. 29, 30), and then select *'Open'* (it is recommended to select the latest updated profile). The profile will load automatically, however, it may take several seconds. The activation of a profile from outside of the software will prevent changing of the default print quality settings.
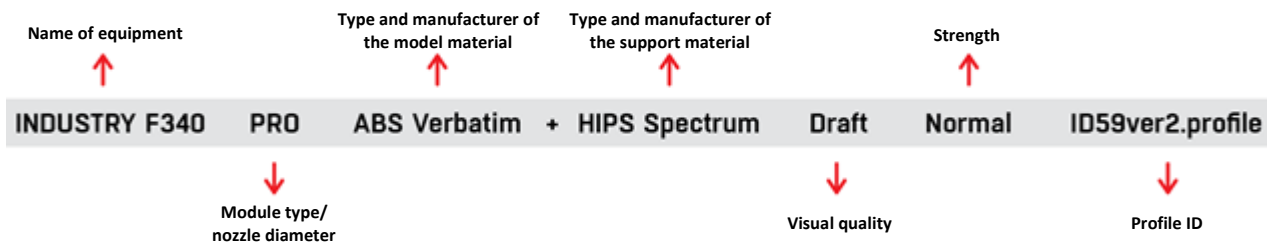
Fig. 29 Description of the name of profile
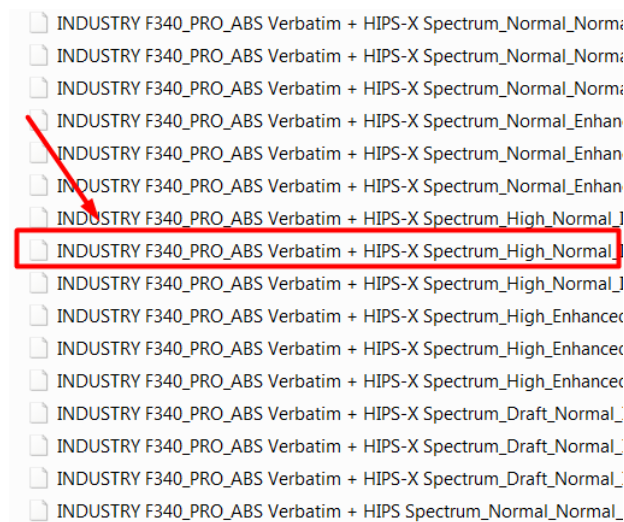
Fig. 30 List of available profile copies

4. After a profile is loaded, the *'Profile* window will show the profile name changed to 'Custom'.
5. Select the *'Prepare to print'* button and wait until the software generates a .gcode or .3dg file.
6. Save the file on your computer with the use of the *'Save to file'* button.

2. EDITING OF ADVANCED PROFILES

Advanced profiles are not connected to the official profiles embedded in the software – the software uses their copies. Thanks to that there is no possibility of accidental damage to the official profiles and they are always available from the standard level of the software interface.

1. Select the *'Expert settings'* option in the *'Adjust print settings'* window or select the option from the top bar.
2. Read and confirm the warning message.
3. From the list, select the profile you want to edit and then select *'Open'* (it is recommended to select the latest updated profile). The profile will load automatically, however, it may take several seconds. The activation of a profile from outside of the software will prevent changing of the default print quality settings.
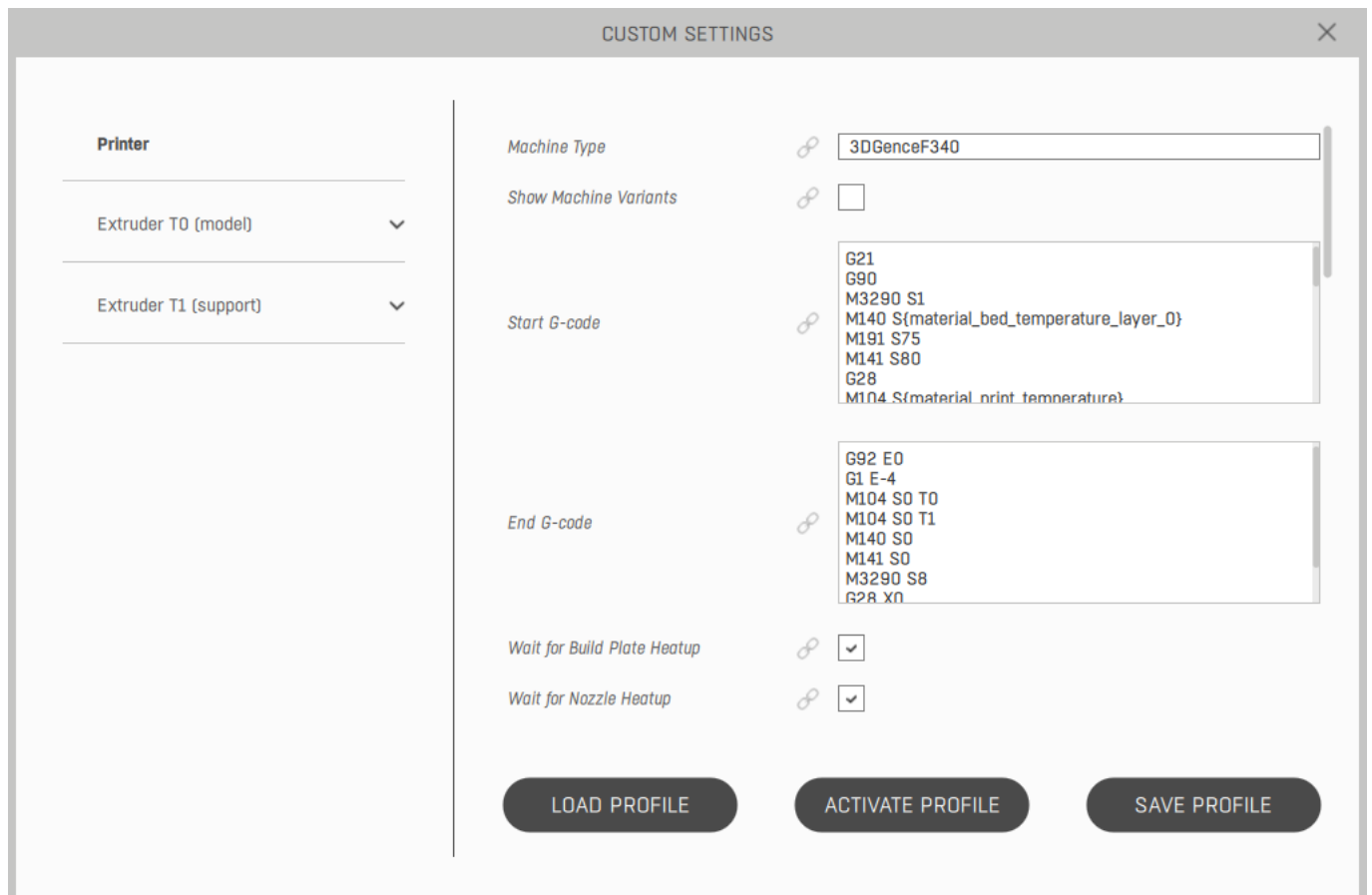4. As soon as the profile is selected, a window appears where settings may be modified (Fig. 31).

Fig. 31 Advance editing window

5. Available buttons:
   - *'Load profile'* – importing of another profile.
   - *'Activate profile'* – activation of a profile without its saving.
   - *'Save profile'* – saving of the profile.

   After selecting the respective button: 'Activate profile' or 'Save profile', and then 'Activate profile', the profile will be activated. The activation of a Custom profile will prevent changing of the default print quality settings.
6. Select the *'Prepare to print'* button and wait until the software generates a .gcode or .3dg file.
7. Save the file on your computer with the use of the *'Save to file'* button.

2.1. Profiles disclosure

The printing profiles generated by way of modification of advanced profiles may be freely disclosed. However, profiles created in various software versions may not be compatible with each other (e.g. a profile generated in version 1.2.0 of the software may not work with the version 1.3.0 or 2.0.1). In such case the print settings must be checked, as there is a risk of damaging the machine or deteriorating the print quality.

Profiles may also be made available to the manufacturer through the notification form on the www.3dgence.com/support website. If the user creates a profile for a non-certified material, actions will be taken to certify the material and enter the same to the official profiles. If the user improves print quality on a material, the manufacturer will take actions to update the official profiles with the changes.

3. MULTIMATERIAL PRINTING

For the advanced users of double-hotend printers, an option of advanced multimaterial printing has been introduced outside of the standard use of the machine. Double-hotend 3DGence printers have been designed and optimised for printing from model and support materials.

The advanced double-hotend printing options enable any choice of materials and selection of the head from which the particular models and their elements will be printed. For example, two-colour print or print of stiff models covered with flexible coat may be made.

3.1. Activation of the advanced two-material printing options

1. Select the double-hotend printer.
2. Load the model for two-material printing consisting of two .stl files.
3. Select the print profile in which the model material is PLA [Verbatim] or PLA [3DGence] and the support material is the same as the model material (Fig. 32). The profile for two-material printing has not been prepared for all double-hotend printers.
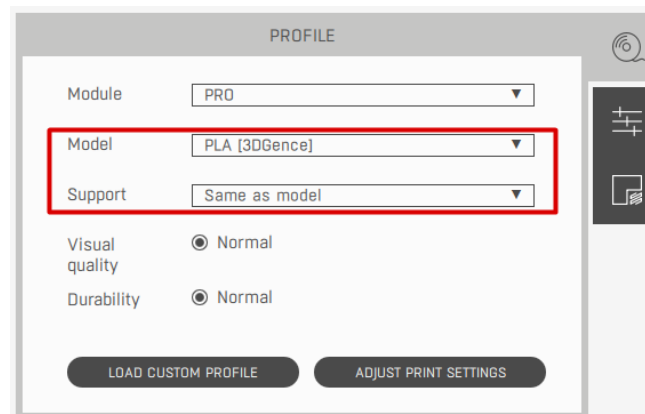

Fig. 32 Profile for two-material printing

4. In the *'Models'* window, select the multimaterial printing option.
5. Read and confirm the warning message.
6. With the use of the function available in the top bar: *'Align selected models'*, set the models in one co-ordinate system.
7. Select T0 material for one of the files and T1 material for the other file. In the *'Models'* window a cursor for selecting the extruder will appear. The '0' position means the print of the selected model with T0 hotend, whereas the '1' position means the print of the selected model with T1 hotend.
8. Select the *'Prepare to print'* button and wait until the software generates a .gcode or .3dg file.
9. Save the file on your computer with the use of the *'Save to file'* button.
10. Load two PLA materials in different colours to the printer and start printing.

VI. SUPPORT AND UPDATES

1. UPDATES

It is recommended to select in the top bar the option of automatic updates and new versions information (Fig. 33). The option may be switched on and off at any moment. Updates are activated each time the software is opened. Additionally, the user may check for the available updates with the use of the *'Check for update'* button. After checking, the respective message will appear on the status bar.
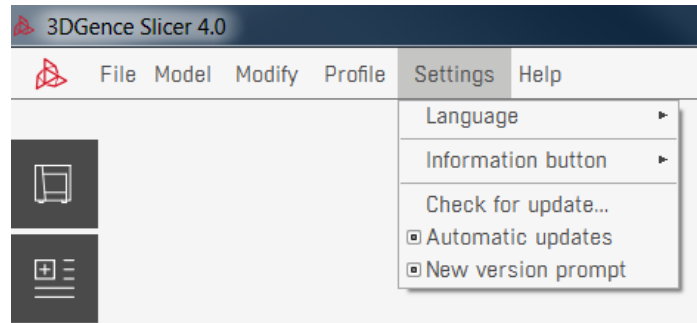

Fig. 33 Top bar – settings

1.1. Print profiles updates

The default material profiles are added to the software by the manufacturer on a current basis. In order to avail of the latest and complete materials database, we recommend switching on of automatic updates, which will enable automatic loading of the latest print profiles to the software (machine and materials settings) provided by the manufacturer. The updates are checked every time when the software is activated, if the user has selected the automatic updates option. Switching off updates completely blocks the attempts of the software contact with any external servers without an explicit user order. The updates require a permit for contact with the manufacturer's servers which is granted to the software by the operating system.

Automatic updates do not affect the existing advanced profiles; however, additional files are loaded in the catalogue with profiles to be edited, identified with a respective date. In order to avail of the older profiles, the profiles with the respective dates must be activated with the use of advanced options.

1.2. Software updates

The 3DGence SLICER 4.0 software updates must be installed manually, using the setup file available on the www.3dgence.com/support website. If the automated updates of print profiles option is selected, the software checks for new versions availability. Several versions of the software may be installed at the same time at one workstation. Print profiles updates are available only for the latest software version.

Despite the fact that the manufacturer applies utmost efforts for the advanced profiles created by the users to be compatible with the latest software versions, print settings must always be checked after an update, as there is a risk of damaging the machine or deteriorating the print quality.

2. SUPPORT

The manufacturer guarantees the possibly highest quality of the models produced with the use of the dedicated software and materials, as well as the prepared default settings. The manufacturer provides also full support in using the print profiles within the software and the recommended printing materials. However, if during the use of the software the user discovers faults in the printed model or software errors, please contact us with the use of the report form available on the www.3dgence.com/support site, with a photo and description of the defect attached and, if possible, a .gcode/.3dg and .stl file (the form is available after an account has been created and the device registered). Each model sent to the manufacturer in that manner will be assessed and/or printed at the company. The manufacturer is going to propose the problem solution by way of an advice, commencement of service activities (if needed), preparing the executable file .gcode/.3dg or print profiles update.

Please remember that the use of materials and print settings which have not been supplied by the manufacturer takes place at the liability of the user. This means that the manufacturer does not provide support regarding the use of advanced profiles.

3DGence Sp. z o.o.
Przyszowice Branch
ul. Graniczna 66, 44-178 Przyszowice
+48 32 438 98 64
support@3dgence.com