Contents

1		What's new?	2
2		Introduction	3
3		Technical Requirements	4
4		Activation	5
	4.1	Offline Activation	5
5	5.1	Updating profiles Export of workspace and profiles	6 6
6		Basic Workflow	7
	6.1	Presets and Profiles	8
	6.2	Model manipulations	9
	6.3	Camera	13
	6.4	Preparing and showing Layers	14
	6.5	Mesh repair	15
7		Terminology	16
8		Troubleshooting	18
8	8.1	Troubleshooting Bed adhesion	18 18
8	8.1 8.2	Troubleshooting Bed adhesion	18 18 18
8	8.1 8.2 8.3	Troubleshooting Bed adhesion	18 18 18 19
8	8.1 8.2 8.3	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics	18 18 18 19 20
8 9	8.1 8.2 8.3 9.1	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects	 18 18 19 20
8 9	8.1 8.2 8.3 9.1 9.2	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects Planned Pauses	 18 18 19 20 23
8 9 10	8.1 8.2 8.3 9.1 9.2	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects Planned Pauses Index of settings	 18 18 19 20 23 25
8 9 10	 8.1 8.2 8.3 9.1 9.2 10.1 	Troubleshooting Bed adhesion Underextrusion Zippers Zippers Advanced Topics Modifier objects Planned Pauses Index of settings Quality Settings: Basic	 18 18 19 20 23 25
8 9 10	 8.1 8.2 8.3 9.1 9.2 10.1 10.2 	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects Planned Pauses Index of settings Quality Settings: Basic Quality Settings: Advanced	 18 18 19 20 20 23 25 29
8 9 10	 8.1 8.2 8.3 9.1 9.2 10.1 10.2 10.3 	Troubleshooting Bed adhesion Underextrusion Zippers Zippers Advanced Topics Modifier objects Planned Pauses Index of settings Quality Settings: Basic Quality Settings: Advanced Quality Settings: Expert - Printer	 18 18 19 20 20 23 25 29 38
8 9 10	 8.1 8.2 8.3 9.1 9.2 10.1 10.2 10.3 10.4 	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects Planned Pauses Index of settings Quality Settings: Basic Quality Settings: Advanced Quality Settings: Expert - Printer Quality Settings: Expert - Layers	 18 18 19 20 20 23 25 29 38 43
8 9 10	 8.1 8.2 8.3 9.1 9.2 10.1 10.2 10.3 10.4 10.5 	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects Planned Pauses Index of settings Quality Settings: Basic Quality Settings: Advanced Quality Settings: Expert - Printer Quality Settings: Expert - Layers Quality Settings: Expert - Support	 18 18 19 20 20 23 25 29 38 43 55
8 9 10	 8.1 8.2 8.3 9.1 9.2 10.1 10.2 10.3 10.4 10.5 10.6 	Troubleshooting Bed adhesion Underextrusion Zippers Advanced Topics Modifier objects Planned Pauses Index of settings Quality Settings: Basic Quality Settings: Expert - Printer Quality Settings: Expert - Layers Quality Settings: Expert - Support Material settings: Basic	 18 18 19 20 23 25 29 38 43 55 63



10.8	Material settings: Expert	73
10.9	Printer settings: Others	76
10.10	Printer settings: Workspace	77
10.11	Printer settings: Motion	81
10.12	Printer settings: Materials	83
10.13	Printer settings: Bed leveling	85
10.14	Printer settings - Custom G-Code	86
10.15	Application	86



What's new?

Version 1.1.1

Features:

- Models are now rendered through the print bed, allowing customers to inspect their model from beneath, both before and after slicing.
- The user can now export a diagnostics file for debugging purpose. The Export Log can be located under Help -> Export Log.
- During slicing, a cancel button will appear. Clicking the button will end the slicing.
- Retract/Pushout settings are now used in the ending GCode. (Changed from initial pushout settings)
- Optimizations of slicing speed in certain situations where objects contains a large number of holes or islands.
- The Slicing window can now be collapsed. During the collapse it is possible to use the layer scroll function if layers are shown.

Bug Fixes:

- Fixed an issue with the "prepare" button appearing active during slicing. The button will now appear faded and present "working" instead of "prepare" during slicing.
- Fixed an issue with the model rendering being slow during scaling of a single model.
- · Fixed an issue with the layer height estimate in the 2D-viewer being calculated incorrectly.
- Fixed an issue with the delete all function not clearing the layer modifiers.



Introduction

Use ION to prepare 3D models for 3D printing. ION can import a multitude of standard CAD formats: STL, PLY, OBJ, and OFF. STL is the industry standard for 3D printing, and that's what we recommend at 3D Labs.

ION is proprietary slicing software built on the robust CreateitREAL commercial slicing engine, specifically designed to work with the 3D Labs X400, and X500 series 3D printers.

ION is tuned for engineering grade and ultra performance high temp materials, and we maintain a list of certified material profiles which can be accessed in the program at any time. We continuously test and certify new materials and publish them.

ION offers a simple workflow, while offering expert settings when needed behind the scenes. Advanced material management capabilities, like speeds per quality setting, chamber heating, pressure advance, and many more are offered in no other slicing software on the market.



Technical Requirements

Operating System:

Windows: Windows 7 or higher, 64 bit

Mac OS / Linux: Not Available - please contact

3D Labs for Alternatives

System Requirements:

Processors: Intel Core i5 4000 or similar

Hard Disk: 1Gb available disc-space

Memory: Minimum 4Gb of RAM



Activation

Upon request you should have received an installer for the software, as well as an activation key that is required to use ION. During the installation you will be asked for activation, which can be done at this moment or later.

If ION is installed without activation, it can be activated later by following these steps:

- 1. Start ION
- 2. Go to "Help" menu
- 3. Click "Activate"
- 4. Click "Retype product key"
- 5. Enter product key and validate

4.1 Offline Activation

For customers that require an airgapped installation (no internet in a highly secure environment), ION can be activated offline.

- 1. Run the ION Offline Installer. Upon completion of the install, go to "Help", then "Activate"
- 2. Call 3D Labs support. You'll need to provide your License ID to the support rep in order to receive the serial number.
- 3. Key in the serial number in ION and press activate.



Updating profiles

ION allows for easy updating of certified material profiles.

5.1 Export of workspace and profiles

ION includes the possibility to export and import workspace files and profiles. Both of these are useful for sharing your work with colleagues, communicating with customer support, and to save your work.

Sliceprofile A sliceprofile includes the settings that are currently applied to the slicing software. The file can be created through the "File" tab, by pressing "Export settings". Likewise, a sliceprofile can be loaded through the same menu.

Workspace file A workspace file includes the objects that are currently loaded in ION, as well as any scaling, rotation, support choices etc. Furthermore, the file contains the current settings applied. This means that a workspace file can recreate exactly the same result in another instance. Workspace files are useful for customer support, as it gives another user the ability to completely replicate your setup. A workspace file has the .rvw file extension, and can be created through the "File" tab by pressing "Save workspace". A workspace file is opened, by selecting "Add file" the same way as an STL file is added.



Basic Workflow

Once ION is installed you are ready to use it with your

3D Labs printer. The basic workflow of ION, from opening a model to exporting a G-Code can be seen below.





- 1. Open File: Start by opening an .STL file.
- 2. Choose printer: Make sure that you select the correct

3D Labs printer from the drop down.

- 3. **Choose extruder:** Select the extruder you'd like to choose the material for. The material will be assigned to this nozzle.
- 4. **Choose material:** Use the dropdown to choose between different materials. For the full list of certified materials, choose "more materials" from any one of the drop down menus.
- 5. **Choose preset** Either choose an existing preset or open settings to adjust the printing options. See section 6.1 for more info
- 6. **Model manipulations:** Set up the workspace using: Move, Scale, Rotate, Mirror, and Support. See section 6.2 for more info.
- Extruder, Objects, Modifiers: Choose which extruder is assigned to the model(s) and supports. Parts will be printed by the nozzle they are assigned to. Drag'n'Drop a model to the modifier area, to make a modifier. This modifier can be used to apply certain settings to certain areas. See section 9.1 for further information.
- 8. **Workspace controls:** reset the camera or the workspace. Resetting the workspace will reload all .STL files in their original scale and default placement.
- 9. **Prepare:** Press "Prepare" when you are ready to create a G-Code file. This button will turn into an "Export" button once preparing is completed.

6.1 Presets and Profiles

ION supports a wide range of materials, through the profiles.

1. Select which

3D Labs Printer you use from the list.

- 2. Choose which extruder you want to select material for.
- 3. Choose material and brand. If the material is not found in the drop down press "More Materials" and select from the complete list of certified materials.
- 4. Select a preset printing profile.



F	3D Labs X400	•	1
	Print head 1	•	2
	ABS		3
	3DXTECH - 3DXMAX		
	Standard	•	4

6.2 Model manipulations

In the following we will go through the model manipulation menu in more detail.

6.2.1 Movement

Movement is the default option in ION

- 1. When this option is selected (which it is pr. default), you can move around objects in the workspace in the XY plane, by Left clicking and dragging.
- 2. By selecting this option, you can move an object on the Z-Axis. Note that moving an object below the workspace will result in only part of the object being printed.
- 3. These fields contain the coordinates of the currently selected model. Values can be added to the field to accurately change an object's location.
- 4. Pressing "Reset" will put all objects back in their default position.



Object Manipulation			
Position	•	2	
X & Y Axis	Z Axi	is	
X: Y: 177.50 17	Z: 77.50	0.00	3
	R	eset	4

6.2.2 Scaling

With these operations you can scale a model up to become larger, or down to become smaller.

- 1. These fields allow you to scale each axis with a certain factor. Scaling an object with a factor of 2 will make it twice as large.
- 2. These fields allows you to scale an object to an absolute size.
- 3. While this box is checked, scaling will happen uniformly on all axes. Unchecking it will allow scaling an object on only one axis.
- 4. Press this button to reset all scaling actions.



Object Manipulation					
Scale			2		
X Scale	1.00	X Size	31.00		
Y Scale	1.00	Y Size	60.00		
Z Scale	1.00	Z Size	48.00		
✓ Fixed	aspect ratio	3	Reset	4	

6.2.3 Rotating

- 1. Choose the number of degrees you wish to rotate your model.
- 2. Rotate the model around the X, Y, or Z Axis.
- 3. Press the reset button to undo all rotations.





6.2.4 Flipping

This allows you to flip or mirror an object along an axis.

- 1. Choose which axis you want to flip or mirror your object along.
- 2. Press the reset button to undo all Flipping.



6.2.5 Support

ION provides three support structure options.

- 1. None: When this option is selected (which it is pr. default) no support is added.
- 2. **Buildplate only:** When this option is selected, support will only be added to overhangs that can be connected directly to the buildplate.
- 3. **Support everywhere** When this option is selected, support will be added everywhere on the model, including places where the support will be placed on the model itself.



3D Labs ION User Manual



6.3 Camera

- 1. Resetting Camera will return the camera to both its original orientation and zoom settings.
- 2. Resetting the workspace will reload all .STL files in their original scale and default placement.





6.4 Preparing and showing Layers

After the Preparing is completed, the layer view will appear.

- 1. Show slices: Toggle between the 3D view of the layers and the default 3D view.
- 2. Pauses: Add a planned pause to the G-Code.
- 3. Visualization: Choose what movements you want to see visualized.
- 4. Slicing info: See information on printing time, and filament consumption.
- 5. **Export:** When ready, press EXPORT to save the G-Code.





6.5 Mesh repair

ION works with STL or OBJ files, both of which can be considered a triangle Mesh. If a triangle mesh is built wrong, then it can be difficult or impossible for the slicer to function correctly. ION will sometimes attempt to repair a model to make it more easily printable. In cases where the mesh repair by ION is not satisfactory, plese refer to the tools below:

For more information about mesh repair visit: https://www.hubs.com/knowledge-base/fixing-most-common-stl-file-errors/

For free mesh repair tools, refer to the below: https://www.meshmixer.com/download.html https://tools3d.azurewebsites.net/



Terminology

The table below provides definitions for the terminology used in ION.



Terminology	Explanation
Concepts	
Printer model	A specific printer, with a set of settings
Filament	Only used when referring specifically to material in "string-
	form" for FDM printers (such as in "filament diameter"). All
	other places "material".
Material	Printing materials in general, which could be referring to
	ABS, PLA or other names and abbreviations of materials.
Profile	A set of settings, containing all the necessary values to cre-
	ate a G-Code file.
Preset (Profile)	A profile defined by
	3D Labs, that cannot be overhuden by the end user. These will be updated when the "Cat newest presents" button in the
	"Indates" tab is pressed
Custom (Profile)	A profile created by the end user. These will stay the same
	after a profile update or a software update but the user is
	prompted to keep them or not
Settings	One or more modifiable values that are used to modify sli-
Cottingo	cing and thus the G-Code created.
Material Profile	The selection of a profile for a certain material.
Material Settings	Settings related to the material.
Printing Profile	Profiles related to the properties of the printed object.
Printing Settings	Settings related to the properties of the printed object.
Object	The 3D printed product made from a 3D model.
(3D) Model	A 3D model. supported 3D model formats includeSTL and
	OBJ. Supported 3D model formats DO NOT include STP,
	DXG etc.
(Model) manipulations	Various changes that can be applied to a 3D model, mainly
	moving, scaling, rotating etc.
XY Axis	The two horizontal Axis of a 3D printer, defined by the home
ZAvia	and the Right hand rule.
Z-AXIS Workopooo (pizo)	The complete group or onvolong that can be printed within
Buildplate	The "bottom" of the workspace. Also known as print bed
Modifier	A model that modifies an object in the intersection between
Modifier	the object and the modifier
Laver	One 3D printed path completed in the XY plane before the
Luyer	Z-Height is increased.
Actions	
Importing (Profile)	Importing is taking a file containing information about the
	profile and loading it into the software, creating a new pro-
	file that has been shared with you by another user, or a sup-
	port team member.
Exporting (Profile)	Exporting a file containing a set of settings, to send to
	someone who wants to replicate these.
Export (G-code or F-Code)	Saving the G-Code or F-Code file generated to a hard drive,
	or similar.
Adding (Profile)	Choosing which of the Profiles are accessible in the Mater-
	ial selection.
Preparing	when a model is transformed into a G-Code or F-Code with
	the chosen settings. This includes, simplification, Disc al-
	iocation, Silcing etc.



Troubleshooting

In this chapter we will cover how to solve some common issues using ION.

8.1 Bed adhesion

If the object does not stick to the print bed, you can try any of these fixes:

- Add brim: go to the "add brim" setting and choose either "always" or "Automatic". See Chapter 10.2.7 for more information
- Increase the temperature of the print bed. Note that this should only be done for certain materials. See Chapter 10.6.1 for more information

8.2 Underextrusion

If the material does not come out of the nozzle in a smooth clean motion, it can have serious consequences for the print quality and object strength.

- Increase the temperature of the print head. See Chapter 10.6.1 for more information
- Make sure that you are using a certified material, and that the material is loaded correctly.



8.3 Zippers

Since a 3D printed object is created one layer at a time, the printer will have to move upwards after completing each layer. This can create a small defect known as a "Seam" or a "Zipper". ION contains settings to manage how this transition happens, to minimize the appearance of this defect.

If the option "hide zippers" is enabled, ION will attempt to put all the small defects in the same place, and hide them in convex or concave angles. If no angles are found that are concave or convex enough, for instance on a cylinder, the zippers will be placed on the back of the model. For certain objects it may be desirable to have the defects be spread around the model, in which case you can disable the "Hide zippers" option. In this case the defects will be placed arbitrarily. See an example below where the left model has hide zippers disabled, and the model on the right has hide zippers enabled. See Chapter 10.2.4 for more information



For more advanced control of the Zippers see: Chapter 10.4.15 for more information.



Advanced Topics

9.1 Modifier objects

Modifier objects allow for changing settings of an object in certain areas, using one or more .STL files to modify another. This has many interesting fields of use, including weight optimization, print time reduction, strength optimization and more.

The recommended workflow for using this feature, is to use a CAD tool to draw zones where you would like the properties to be different. Below will follow an example of how to use this feature for creating a frame that has a higher infill density in the corners:

Step 1: In the picture below, a frame is designed (shown in blue) with four boxes in the corners (shown in white). The four corners are grouped together, and the frame and the corners are exported as individual .STL files.





Step 2: In the picture below the two .STL files are seen loaded into ION.



Step 3: Select both models by holding down SHIFT and left clicking them. Afterwards align and group them using the function in the "edit" tab, or the shortcut CTRL + SHIFT + j.



Step 4: Drag'n'drop the 3D model from the object area, into the modifier area. Now the modifier menu will become available for the modifier object.



3D Labs ION User Manual



Step 5: You now have access to the modifier menu, where you can change the settings for the modifier. This will change the settings in all the places where the modifier overlaps the model.

Change modifiers		ModifierSettingsWindow	man to and a second of
⊙ Objects		Infill	
frame		Weak Strong	Infill density 55%
	THE FLETCHER STATE	Infill type:	3D Gyroid Lattice 💙
		Infill line width (mm):	0.45 😴
 Modifiers 	ATATATATA		
Corners	ALALAAAA		
\$ •			
	ALT ALT ALT ALT A		

Step 6: Below the result can be seen, where the corners have a 55% infill density, whereas the rest of the model has 5%.





9.2 Planned Pauses

ION includes a feature to insert planned pauses into your print process. This can be used to create a part that changes color after a certain layer, or to embed various objects into a 3D printed object. Here follows an example of how to embed a nut into a 3D printed object.

Step 1: Designing the 3D object When designing the CAD model you need to make space for the nut inside.



Step 2: Preparing model in ION. Load the .STL file into ION and prepare it. Now open the slices menu, and navigate to the layer where the nut will be completely embedded into the object. On the pictures it can be seen that the pause needs to be between layer 25 and 26. Press either the "Add pause after layer" or "Add pause before layer" - button. Now a pause has been added, and your printer will pause at this place.





Step 3: Printing. When the pause appears in the menu as seen in the picture below, you are ready to start the print. The printer will automatically pause between these two layers and prompt the user to take action.



Step 4: Finished product. When the product is finished, a bolt can be inserted into the nut.





Index of settings

10.1 Quality Settings: Basic

To open the Quality settings, press the settings button as showed below:



10.1.1 Layer Resolution

Placement	Setting Name	Illustration
Basic	Layer resolution	

Layer thickness (mm):

- Determines the height of each layer.
- Thinner layers can improve the surface quality of detailed objects, but will increase the print time.



10.1.2 Infill Density

Placement	Setting Name	Illustration
Basic	Infill density	

Infill density:

- Higher infill will result in a stronger object, at the cost of material usage and print time.

- 100% will result in a solid object, and 0% will result in a hollow object.

10.1.3 Shell thickness

Placement	Setting Name	Illustration
Basic	Shell thickness	

Shell thickness:

- Defines the thickness of the solid outer shell of the object.

- It is defined as a multiple of the line width.

- A higher value will result in a stronger object at the cost of material usage and print time.

10.1.4 Infill type

Placement	Setting Name	Illustration
Basic	infill type	

ION includes several different **Infill types** listed below. Note that some of them are 3D infills. These are geometries that are generated to be different for each layer. These have different interesting properties, for instance the Gyroids and diamond infills provide a high tensile strength while the Isomax provides a stiff geometry.



3D Labs ION User Manual

- Linear infill
- Crossed infill
- 3D Diamond
- 3D Double Gyroid
- 3D Gyroid latice
- 3D Gyroid Skeleton
- 3D Isomax
- 3D Truncated Octahedron
- Fish
- Honeycomb
- Morrocan Star
- Octagonal
- Triangular

Find on the next page an overview of the properties of the 3D infills:



3D infill	Properties
3D diamond	- Evens out mechanical strength on all directions
	- Reduces print time by up to 10% for infill densities smaller than 15% *
	- Reduces object weight and material use
	- Surface area and channel connectivity properties can be used for a
	variety of applications: from tissue scaffolds, biofilters, metamaterials
	for thermal and sound insulation or photonic manipulation
	https://www.sciencedirect.com/science/article/pii/
	S0032386117311175
Double Gyroid	- Evens out mechanical strength on all directions
	- Large surface area
	- Surface area and channel connectivity properties can be used for a
	variety of applications: from tissue scaffolds, biofilters, metamaterials
	for thermal and sound insulation or photonic manipulation
	https://www.sciencedirect.com/science/article/pii/
	S0032386117311175
Gyroid lattice	This structure optimizes object strength for stress in all directions. This
	keeps the object weight minimal and printing speed high.
	- Reduces material use and object weight
	- Evens out mechanical strength on all directions
	- Increases mechanical strength
	- High degree of channel interconnectivity
	- Surface section has no straight lines
	- No planes of symmetry
	- Surface area and channel connectivity properties can be used for a
	variety of applications: from tissue scaffolds, biofilters, metamaterials
	for thermal and sound insulation or photonic manipulation
	https://www.sciencedirect.com/science/article/pii/
	S0032386117311175
Gyroid skeleton	- Evens out mechanical strength on all directions
	- Reduces object weight and material use
	- Surface area and channel connectivity properties can be used for a
	variety of applications: from tissue scaffolds, biofilters, metamaterials
	for thermal and sound insulation or photonic manipulation
3D isomax	This structure optimizes object toughness and crush-resistance by us-
	ing a structure that reaches the theoretical limits of elastic stiffness
	while also having a very low weight. The isomax also evens out strength
	in all directions.
	- Big increase of mechanical strength
	- Evens out mechanical strength on all directions
	https://www.nature.com/articles/s41467-020-15434-2
3D truncated Octahedron	A type of 3D honeycomb structure. Similar to the gyroid and the isomax,
	the truncated octahedron evens out mechanical strength in objects.



10.2 Quality Settings: Advanced

10.2.1 Retraction enable/disable

Placement	Setting Name	Illustration
Advanced	Retraction enable/disable	

Retraction is a feature to reduce dripping of excess plastic during repositioning. Before repositioning, the extruder sucks back the material to avoid drops. By the end of the repositioning, the material is pushed back to its original position and the printing continues as normal.

10.2.2 Continous Z

Placement	Setting Name	Illustration
Advanced	Continous Z	
)))))))))))))))))))))))))))))))))))))))

Continuous Z:

- Continuous movement on Z axis during the entire print can increase wall quality by smoothing out the layer change if there are no repositioning within a layer.

- Use with care. This setting can create uneven layers and bad quality for complex objects and objects with repositioning within a layer.

- This setting will have very little effect on objects printed with infill and more than one contour.



10.2.3 3D shell

Placement	Setting Name	Illustration
Advanced	3D shell	

3D Shell:

- Creates a 3D shell within the object improving object's strength, helping closing open areas with visible infill and allowing better free forming ("building in the air").

10.2.4 Hide zippers

Placement	Setting Name	Illustration
Advanced	Hide zippers	

Hide zippers:

The very start of a layer will often have very small defects called zippers. When this checkbox is checked, these defects will be hidden if possible.



10.2.5 Preserve internal geometry

Placement	Setting Name	Illustration
Advanced	Preserve internal geometry	

Preserve internal geometry:

- Many 3D models are created as a combination of smaller objects intersecting each other. This results in unwanted internal geometries that are automatically removed during slicing.

- Use this option if you want to preserve the inside of geometry the models. For instance, if you have an 3D model that is designed to be a hollow object, but the inside is automatically filled during slicing.

10.2.6 Disable simplification

Placement	Setting Name	Illustration
Advanced	Disable simplification	

Disable simplification:

To decrease slicing time 3D models are simplified when opened.

Disable simplification if you experience that you have lost details on your object after slicing. Note that this will increase the slicing time significantly.

10.2.7 Print brim

Placement	Setting Name	Illustration
Advanced	Print brim	



Print brim:

- This builds an extra wide bottom layer by adding extra, easily removable contours.
- The maximum number of contours that fit inside this distance, will be generated.
- This is useful if the object has trouble sticking to the bed.
- If "Automatically" is selected, the shapes of the objects determine the brim settings.

10.2.8 Fill thin walls

Placement	Setting Name	Illustration
Advanced	Fill thin walls	

Fill thin walls: Will attempt to fill up thin parts of the object that is normally too small to print. In some cases this can make parts of the print slightly thicker than expected. There are three options:

- None: Areas will only have infill if there is sufficient room for the filament. Use this if you have strict requirements for the dimension tolerances of your object
- Only visible: Only visible areas will be allowed to be filled extra up.
- All: The whole print is allowed to be filled extra up.

10.2.9 Infill height

Placement	Setting Name	Illustration
Advanced	Infill frequency	

Infill height (mm):

Defines the thickness for infill and the distance between layers with infill. Full infill areas are not affected



and will be printed on each layer.

- This allows faster prints, by only printing infill on some layer, without decreasing the quality of visible areas.

- If the distance is the same as the layer thickness, there is infill on every layer.

- The "Infill frequency" value has an upper limit of 1.5 * "Nozzle diameter".

10.2.10 Solid layers: Top thickness

Placement	Setting Name	Illustration
Advanced	Solid layers: Top thickness	

Top thickness:

-Determines the amount of solid infill layers at the top shell of the object.

-Increasing the top thickness can improve the quality and strength of objects at the cost of material usage and print time.

10.2.11 Solid layers Bottom thickness

Placement	Setting Name	Illustration
Advanced	Solid layers Bottom thickness	

Bottom thickness:

-Determines the amount of solid infill layers at the bottom shell of the object.

-Increasing the bottom thickness can improve the quality and strength of objects at the cost of material usage and print time.



10.2.12 Visible areas: Speed

Placement	Setting Name	Illustration
Advanced	Visible Areas Speed	
		[mm/s]↑
		Max.
		Min
		→ →

Visible area speed (mm/s):

- The target maximum printing speed for visible areas.

10.2.13 Visible areas: Acceleration rate

Placement	Setting Name	Illustration
Advanced	Visible Areas Acceleration rate	
		[mm/s] Max. Min.

Acceleration rate for printing visible areas (mm/s²):

- Defines the rate with which the speed changes during printing of the visible areas.

10.2.14 Inner areas: Speed

Placement	Setting Name	Illustration
Advanced	Inner areas: Speed	

Inner area speed factor (mm/s):

- Determines the target speed while printing inner areas.



- Lowering this value may improve the quality of the object, at the expense of increasing the printing time.

- Increasing this value will decrease the printing time, but may result in bad quality.

- If a red "!" appears next to the value, the target speed get limited by the maximum extrusion. This means that the speed will not be reached, to avoid under-extrusion.

10.2.15 Support: Overhang angle

Placement	Setting Name	Illustration
Advanced	Support: Overhang angle	

Overhang angle (degrees):

- Maximum allowed angle for overhangs without building support.
- 0° = vertical, 90° = horizontal line with respect to the build platform.
- Check the "Show layers" window, to see if the support is applied at the desired places.

10.2.16 Support: infill density

Placement	Setting Name	Illustration
Advanced	Support: infill density	

Support Density:

- Density of of the support.

- 100% will print solid support layers, 0% will not make any infill.

- High support densities will create a more stable support, which will give better support under overhangs, but will be more difficult to remove and increase the print time.


10.2.17 Support: Z-Offset top

Placement	Setting Name	Illustration
Advanced	Support: Z-Offset top	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

10.2.18 Support: Z offset bottom

Placement	Setting Name	Illustration
Advanced	Support: Z offset bottom	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

10.2.19 Object type Standard

Placement	Setting Name	Illustration
Advanced	Object type Standard	



Object Types :

- Closed (Left): The object will print as shown in the workspace.

- Vase (Center): The object will print without infill and without solid top layers. Should be used if the model has a solid top and infill.

- Bracelet (Right): The object will print without infill, solid top and bottom layers. Should be used if the model has a solid top and bottom and infill.

10.2.20 Object type Vase

Placement	Setting Name	Illustration
Advanced	Object type Vase	

Object Types :

- Closed (Left): The object will print as shown in the workspace.

- Vase (Center): The object will print without infill and without solid top layers. Should be used if the model has a solid top and infill.

- Bracelet (Right): The object will print without infill, solid top and bottom layers. Should be used if the model has a solid top and bottom and infill.

10.2.21 Object type Bracelet

Placement	Setting Name	Illustration
Advanced	Object type Bracelet	

Object Types :

- Closed (Left): The object will print as shown in the workspace.

- Vase (Center): The object will print without infill and without solid top layers. Should be used if the model has a solid top and infill.

- Bracelet (Right): The object will print without infill, solid top and bottom layers. Should be used if the model has a solid top and bottom and infill.



10.3 Quality Settings: Expert - Printer

10.3.1 Printing speed

Placement		Setting Name	Illustration
Expert	Printer	Printing speed	[mm/s] Max. Min.

Maximum printing speed (mm/s):

- Defines the target printing speed.

- The maximum value of the "Speed Factor" sliders, the Advanced tab are defined by this setting.

- When the "Speed Mode" slider in the Advanced tab are disabled, this value is the target maximum printing speed for inner areas. Disable these sliders with the checkbox below.

- Note that this value might not be reached if Maximum extrusion is too low.

10.3.2 Visible area speed

Placement		Setting Name	Illustration
Expert	Printer	Visible area speed	[mm/s] Max. Min.

Visible area speed (mm/s):

- The target maximum printing speed for visible areas.



10.3.3 Repositioning speed

Placement		Setting Name	Illustration
Expert	Printer	repositioning speed	
			c (C)
			B OTTOM

Repositioning speed (mm/s):

- The target maximum speed for repositioning between printing paths.

10.3.4 Z-Only repositioning speed

Placement		Setting Name	Illustration
Expert	Printer	Z-Only repositioning speed	

Repositioning speed (mm/s):

- The target maximum speed for repositioning between printing paths.

10.3.5 Enable impulse and acceleration

Placement	Setting Name	Illustration
Expert	Enable impulse and acceleration	
		[mm/s] Max. Min.

Enable Impulse and Acceleration control in the G-Code file: If checked, ION will generate acceleration and impulse G-Codes.



If left unchecked, the printer firmware will use its own acceleration and jerk values. In this case print time estimation may be inaccurate.

10.3.6 Acceleration rate: Printing

Placement		Setting Name	Illustration
Expert	Printer	Acceleration rate: Printing speed	
			[mm/s] Max. Min.

Acceleration rate for printing (mm/s^2):

- Defines the rate with which the speed changes during printing of the inner areas.

10.3.7 Acceleration rate: Visible areas

Placement		Setting Name	Illustration
Expert	Printer	Acceleration rate: Visible areas	
			[mm/s] Max. Min.

Acceleration rate for printing visible areas (mm/s²):

- Defines the rate with which the speed changes during printing of the visible areas.

10.3.8 Acceleration rate: Repositioning

Placement		Setting Name	Illustration
Expert	Printer	Acceleration rate: Repositioning	
			B CONTRACTOR



Acceleration Rate & Deceleration Rate for repositioning (mm/s²):

- Specifies the speed change during repositioning.

10.3.9 Acceleration rate: Z

Placement		Setting Name	Illustration
Expert	Printer	Acceleration rate: Z	

Acceleration Rate & Deceleration Rate for repositioning (mm/s²):

- Specifies the speed change during repositioning.

10.3.10 Impulse: visible area

Placement		Setting Name	Illustration
Expert	Printer	Impulse: visible area	[V] 150 100 50 [†]
			[[]

Impulse speed (mm/s):

- The allowed jump in speed on a motor axis.

- This value is used to assign speed limits on path corners.

- All movements have initial speeds based on the impulse speed of the moving axes and the direction of the first movement.

- Too high impulse speed values can cause ringing defects on the print surface.



10.3.11 Impulse Other

Placement		Setting Name	Illustration
Expert	Printer	Impulse Other	[v] †
			150 100 50
			[t]

Impulse speed (mm/s):

- The allowed jump in speed on a motor axis.

- This value is used to assign speed limits on path corners.

- All movements have initial speeds based on the impulse speed of the moving axes and the direction of the first movement.

- Too high impulse speed values can cause ringing defects on the print surface.

10.3.12 Impulse: Impulse Z-Axis

Placement		Setting Name	Illustration
Expert	Printer	Impulse: Impulse Z-Axis	[V] 150 100 50 [t]

Impulse speed (mm/s):

- The allowed jump in speed on a motor axis.

- This value is used to assign speed limits on path corners.

- All movements have initial speeds based on the impulse speed of the moving axes and the direction of the first movement.

- Too high impulse speed values can cause ringing defects on the print surface.



10.3.13 Use repositioning Hop

Placement		Setting Name	Illustration
Expert	Printer	Use repositioning Hop	

Repositioning hop:

- During repositioning, the head will move away from the bed.

- This feature is used to avoid leaving marks when crossing contours.

10.4 Quality Settings: Expert - Layers

10.4.1 Layer Thickness

Placement		Setting Name	Illustration
Expert	Layers	slice thickness	

Layer thickness (mm):

- Determines the height of each layer.

- Thinner layers can improve the surface quality of detailed objects, but will increase the print time.

- To enable this value, select "Custom" as Layer resolution in the Basic tab.



10.4.2 Number of contours

Placement		Setting Name	Illustration
Expert	Layers	Shell thickness	

Number of contours:

- Number of lines in a wall.

- Increasing the number of contours can improve strength and print quality.

10.4.3 Contour width

Placement		Setting Name	Illustration
Expert	Layers	contour width	

Contour width:

- The width of the printed lines for contours.
- This value changes automatically when the value of the setting "Nozzle diameter" is changed.

10.4.4 Contour width smallest detectable

Placement		Setting Name	Illustration
Expert	Layers	contour width smallest detect- able	



Contour width smallest printable/detectable:

- When printing very small details that cannot be printed with regular contour line width, the smallest detectable line width is applied.

- These details are printed with the smallest printable contour line width.

- The smallest printable contour width should be bigger than the smallest detectable contour width to avoid underextrusion for very slow extrusion.

10.4.5 Contour width smallest printable

Placement		Setting Name	Illustration
Expert	Layers	contour width smallest printable	

Contour width smallest printable/detectable:

- When printing very small details that cannot be printed with regular contour line width, the smallest detectable line width is applied.

- These details are printed with the smallest printable contour line width.

- The smallest printable contour width should be bigger than the smallest detectable contour width to avoid underextrusion for very slow extrusion.

10.4.6 Contour overlap

Placement		Setting Name	Illustration
Expert	Layers	Contour overlap	

Contour overlap:

- How much of the contour overlaps the neighboring contours.

- 100% means that half of the contour overlaps the contour to the right and the other half overlaps the contour to the left.

- 0% means that the contours don't overlap each other.



10.4.7 Infill line width

Placement		Setting Name	Illustration
Expert	Layers	Infill line width	

The width of the printed filament line for infill. This value changes automatically when Nozzle diameter (10.12.1) is changed.

10.4.8 Infill overlap

Placement		Setting Name	Illustration
Expert	Layers	Infill overlap	

Determines how much the infill lines may overlap the inner contour. 50% means that the the infill may overlap half of the width of the inner contour. If the contours are not completely attached to the infill, increase this value. This can sometimes be necessary when using flexible filaments.

10.4.9 Infill density (same as basic)

Same as Section 10.1.2

Placement		Setting Name	Illustration
Expert	Layers	Infill density (same as basic)	



10.4.10 Solid infill threshold area

Placement		Setting Name	Illustration
Expert	Layers	solid infill threshold area	

Areas smaller than this value will be using 100% infill density to ensure rigidity of the object.

10.4.11 infill angle

Placement		Setting Name	Illustration
Expert	Layers	infill angle	

Determines the angle the infill lines. The angle only affects Crossed and Linear infill. Typical infill values are 45 or 0 degrees.

10.4.12 Visible infill extrusion ratio

Placement		Setting Name	Illustration
Expert	Layers	Visible infill extrusion ratio	



Determines the extrusion rate for visible infill lines such as local top layers. Use this to compensate for cases when there is visible space between the infill lines off visible areas.

10.4.13 Generate automatic infill connections

Placement		Setting Name	Illustration
Expert	Layers	Generate automatic infill con-	None
		nections	

If enabled, ION will try to make sure infill lines are connected at the ends. If disabled, you can specify the length of additional infill before and after infill lines. These will follow the shape of the contour. The distance of these connecting lines can be defined in the field "Leading/trailing infill line length" 10.4.14.

10.4.14 Leading/trailing infill line width

This field is only available if "Generate automatic infill connections" 10.4.13 is disabled, and in that case gives the ability to control a trail before the infill line.

Placement		Setting Name	Illustration
Expert	Layers	Leading/trailing infill line width	None

Specifies the distance that infill should not go above when trying to connect infill lines. If the specified length is not possible to achieve, the closest possible length will be used. Set this to 0 to remove all infill connections.



10.4.15 Hide zippers

Placement		Setting Name	Illustration
Expert	Layers	Hide zippers	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.

2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.

3) Put them on the back of the object.

10.4.16 Hide in concave angles

Placement		Setting Name	Illustration
Expert	Layers	Hide in concave angles	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.

2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.

3) Put them on the back of the object.



10.4.17 Hide in convex angles

Placement		Setting Name	Illustration
Expert	Layers	Hide in convex angles	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.

2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.

3) Put them on the back of the object.

10.4.18 Hide on the back

Placement		Setting Name	Illustration
Expert	Layers	Hide on the back	

Hide Zippers:

Hiding the zippers will be prioritized in the order they appear:

1) Hide them inside concave sharp angles if possible. A concave angle points into the object. Specify the biggest acceptable angle in degrees.

2) Hide them inside convex sharp angles if possible. A convex angle points out of the object. Specify the acceptable interval of angles in degrees.

3) Put them on the back of the object.



10.4.19 Stitch Seam

Placement		Setting Name	Illustration
Expert	Layers	Stich Seam	

Stitch seams:

If enabled, the print head will move slightly into the object after finishing the contour in order to fuse the start and end of the path.

10.4.20 Contour printing strategy

Placement		Setting Name	Illustration
Expert	Layers	Contour printing strategy	

Contour printing strategy:

Control the printing order of contours. The options are:

- Inner first - this will print the inner contours before the outer contours

- Outer first - this will print the outer contours, and then the inner

Note: Printing outside contours first can decrease print quality on the outer surfaces, especially on areas with overhangs.



10.4.21 Repositioning strategy

Placement		Setting Name	Illustration
Expert	Layers	Repositioning strategy	
			C
			В •

Repositioning strategy:

Define which path finding strategy is used during slicing of the model(s)

- Support first - A strategy which ensures that support structures are printed first on every layer

- Closest distance - A strategy which looks for the closest distance to the printhead

10.4.22 Avoid printed parts

Placement		Setting Name	Illustration
Expert	Layers	Avoid printed parts, Avoid dis-	
		tance	
			C
			В

"Avoidance distance" is only available if "Avoid printed parts" 10.4.22 is enabled.

Avoid printed parts:

Avoid printed parts when repositioning between areas by moving around them.

Avoid distance (mm):

- The distance from the printed parts to the repositioning path.

- This value changes automatically when the value of the setting "Nozzle diameter" is changed.



10.4.23 Brim offset

Placement		Setting Name	Illustration
Expert	Layers	Brim offset	

Print brim:

- This builds an extra wide bottom layer by adding extra, easily removable contours.
- The maximum number of contours that fit inside this distance, will be generated.
- This is useful if the object has trouble sticking to the bed.
- If "Automatically" is selected, the shapes of the objects determine the brim settings.

10.4.24 Print internal brim with offset

Placement		Setting Name	Illustration
Expert	Layers	Print internal brim with offset.	

Print internal brim:

- Adds a brim inside holes of objects and objects inside holes.
- The maximum number of contours that fit inside this distance, will be generated.
- This is useful if the printed object have trouble sticking to the bed.
- Brim width is defined as the distance from the edge of the object to the edge of the brim.
- Internal brim isn't illustrated in the workspace. Use show layers to check if the brim is as intended.
- Using internal brim with objects that require internal support may cause overlap.



10.4.25 Print skirt with Offset

Placement		Setting Name	Illustration
Expert	Layers	Print skirt with Offset	

Print skirt:

- Build an offset line of material around the object on the first layer to ensure that the printhead is ready to print.

- Printing a skirt can improve bed adhesion.

10.4.26 Initial push-out speed

Placement		Setting Name	Illustration
Material	Advanced	Initial push-out speed	[mm/s]

Initial push-out speed:

- Speed of the initial push-out

- This is also the speed used for manual extrusion in the Printing window.



10.5 Quality Settings: Expert - Support

10.5.1 XY offset

Placement		Setting Name	Illustration
Expert	Support	XY offset	

XY offset (mm):

- Distance between the support and the printed object on X and Y axes

10.5.2 Z-Offset top layer

Placement		Setting Name	Illustration
Expert	Support	Z-Offset top layer	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.



10.5.3 Z-Offset bottom layers

Placement		Setting Name	Illustration
Expert	Support	Z-Offset bottom layers	

Z-offset (mm):

- Distance between the support and the printed object on the Z axis.

10.5.4 Overhang angle

Placement		Setting Name	Illustration
Expert	Support	Overhang angle	

Overhang angle (degrees):

- Maximum allowed angle for overhangs without building support.
- 0° = vertical, 90° = horizontal line with respect to the build platform.
- Check the "Show layers" window, to see if the support is applied at the desired places.

10.5.5 Support: Infill density

Placement		Setting Name	Illustration
Expert	Support	Infill density	



Support Density:

- Density of of the support.
- 100% will print solid support layers, 0% will not make any infill.

- High support densities will create a more stable support, which will give better support under overhangs, but will be more difficult to remove and increase the print time.

10.5.6 Support: Full infill density

Placement		Setting Name	Illustration
Expert	Support	Full Infill density	

Full support density (%):

-Defines the density of the solid support layers used at the local tops of the support. -100% will print solid support layers, 0% will not make any infill.

10.5.7 Support: Infill angle

Placement		Setting Name	Illustration
Expert	Support	Infill angle	

Infill angle:

- Determines the angle the support infill lines.
- Typical support infill are 45 or 0 degrees. 45 degree is recommended.



10.5.8 Number of contours

Placement		Setting Name	Illustration
Expert	Support	Number of contours	

Number of support contours:

- Number of lines in a support contour / wall.

- Increasing the numbers of contours can improve the stability of the support.

10.5.9 Over/under extrusion ratio

Placement		Setting Name	Illustration	
Expert	Support	Over/under extrusion ratio		
			1 00%	↓100%

Over/under-extrusion ratio (%):

- Set over/under-extrusion ratio to a value greater than 100% to over-extrude and less to under-extrude.

- Under-extrusion is beneficial for support as it results in a brittle support that is easier to remove.



10.5.10 Top/bottom over/under extrusion

Placement		Setting Name	Illustration	
Expert	Support	Top/bottom over/under extru- sion	† 100%	L 100%
				• 100%

Over/under-extrusion ratio Top/Bottom (%):

- Set over/under-extrusion ratio to a value greater than 100% to over-extrude and less to under-extrude.

- 100% or slightly over-extruded is beneficial, since this is the part that sticks to the bottom.

10.5.11 Crossed infill

Placement		Setting Name	Illustration
Expert	Support	Crossed infill	

Crossed support:

- Support infill will be printed in the crossed infill pattern.

10.5.12 Support Contour Width

Placement		Setting Name	Illustration
Expert	Support	Support contour width	None

Support contour width:



- The width of printed support contours.

- This value changes automatically when the value of the setting "Nozzle diameter" is changed.

10.5.13 Support Contour overlap

Placement		Setting Name	Illustration
Expert	Support	Support contour overlap	None

Support contour overlap:

- How much of the contour overlaps the neighboring contours.

- 100% means that half of the contour overlaps the contour to the right and the other half overlaps the contour to the left.

- 0% means that the contours don't overlap each other.

10.5.14 Support Contour overlap smallest detectable

Placement		Setting Name	Illustration
Expert S	Support	Contour overlap smallest detect- able	None

Support contour width smallest detectable:

- When printing very small details, which require support that cannot be printed with regular support contour width, the smallest detectable width is applied.



10.5.15 Support top thickness

Placement		Setting Name	Illustration
Expert	Support	Number of support top layers	

Support top thickness (mm):

- Defines the distance from the top of a support structure with solid support infill.

10.5.16 Support bottom thickness

Placement		Setting Name	Illustration
Expert	Support	Number of support bottom lay-	
		ers	

Support bottom thickness (mm):

- Defines the distance from the bottom of a support structure with solid support infill.

10.5.17 Support infill distance

Placement		Setting Name	Illustration
Expert	Support	Support infill frequency layers	



Support Infill distance (mm):

Defines the layer thickness for support infill. Full infill areas are not affected and will be printed on each layer.

- This allows faster prints, by printing less support infill.
- If the distance is the same as the layer thickness, there is infill on every layer.
- Infill frequency has upper limit to 1.5 x Nozzle diameter

10.5.18 Remove holes smaller than

Placement		Setting Name	Illustration
Expert	Support	Remove holes smaller than	

Remove holes in the support with an area smaller than the specified value

10.5.19 Remove holes smaller than in 3d

Placement		Setting Name	Illustration
Expert	Support	Remove holes smaller than in 3d	
			[mm]

Remove holes in the support with a width less than the specified value



10.5.20 Generate automatic infill connections

Placement		Setting Name	Illustration
Expert	Support	Generate automatic infill con-	None
		nections	

If enabled, ION will try to make sure infill lines are connected at the ends. If disabled, you can specify the length of additional infill before and after infill lines. These will follow the shape of the contour. The distance of these connecting lines can be defined in the field "Leading/trailing infill line length" 10.5.21.

10.5.21 Leading/trailing infill line width

This field is only available if "Generate automatic infill connections" 10.5.20 is disabled, and in that case gives the ability to control a trail before the infill line.

Placement		Setting Name	Illustration
Expert	Support	Leading/trailing infill line width	None

Specifies the distance that infill should not go above when trying to connect infill lines. If the specified length is not possible to achieve, the closest possible length will be used. Set this to 0 to remove all infill connections.

10.6 Material settings: Basic

To open the Material settings:

- 1. Press "settings"
- 2. Open "Material settings override"



3D Labs ION User Manual

Ĩ	3D Labs X400	•	%	2	Printing settings Basic settings Advanced settings Expert settings		Tooltip	``	`
	Print head 1	•			- Global layer thickness Standard × Layer thickness 0.2 mm	25	a Material settings Basic settings Advanced	settings Expert settings	
6	ABS	*	°,		Object Strength Weak Strong Infil density 20%		Material Manufacturer: Material type:	3DXTECH ABS	
F	Standard	-	1		Shell thickness 0.8 mm 2 contours Infill type: 2 Crossed Infill v		Material density (g/cm ²): Print head Printing temperature (*C)	:	250
23			0.9		Override material settings		Heated Bed		

10.6.1 Print head temperature

Placement		Setting Name	Illustration
Material	Basic	Print head temperature	

Printing temperature:

- The temperature at which the material is melted in the printhead.

10.6.2 Print bed temperature

Placement		Setting Name	Illustration
Material	Basic	Print bed temperature	

Heated bed temperature (°C):

- The temperature of the heated bed while printing.
- If you have problems with objects not sticking to the bed, consider raising the temperature.



10.7 Material settings: Advanced

10.7.1 Maximum extrusion

	Setting Name	Illustration
vanced	Maximum extrusion	
	vanced	vanced Maximum extrusion

Maximum extrusion (mm³/s):

- Defines the maximum amount of material per second that is allowed to be pushed out of the print head.

- Value is dependent on the material type, since different materials melts at different temperatures.

- This setting will limit the maximum printing speed, if necessary.

Tip: If you experience under-extrusion try to decrease this value.

10.7.2 Min time pr layer

Placement		Setting Name	Illustration
Material	Advanced	Min time pr layer	

Minimum time per layer in seconds:

- Defines the minimum allowed time for a layer to be printed.

- Slows down the speed at small layers in order to have enough time to cool and harden the material.
- Dynamically increases the fan speed to cool down the material.
- Considerably improves the quality of details of small layers.
- To bypass, enter 0 as value.



10.7.3 Cooling

Placement		Setting Name	Illustration
Material	Advanced	Maximum fan speed	
			$\overbrace{FAN 3}^{(2)} \swarrow \overbrace{fan 4}^{(2)} \swarrow \overbrace{fan 4}^{(2)} \swarrow \overbrace{fan 4}^{(2)} \swarrow \overbrace{fan 4}^{(2)} \swarrow \overbrace{fan 2}^{(2)} \checkmark \overbrace{fan 2}^{(2)} \checkmark \overbrace{fan 2}^{(2)} $

MISSING

10.7.4 Maximum fan speed

Placement	Setting Name	Illustration
Material Advance	ed Maximum fan speed	$FAN 3 \qquad $

Maximum fan speed:

This value determines the maximum speed that the fan is allowed to reach.

If this value is higher than the target fan speed, ION will dynamically increase the fan speed towards the maximum allowed speed, if a layer is slowed down, due to Minimum time per layer violations. In order to deactivate this feature, set the default and maximum fan speed to the same value.



10.7.5 Initial pushout

Placement		Setting Name	Illustration
Material	Advanced	initial push-out	[mm]

Initial push-out (mm):

The length of material pushed into the printhead before printing. The initial push-out will happen as soon as the target temperatures are reached.

- This is done in order to ensure that there is material in the printhead.

10.7.6 Custom first layers: Number of layers

Placement		Setting Name	Illustration
Material	Advanced	Custom first layers: Number of	
		layers	

Number of bottom layers:

- Choose the number of bottom layers that will use 100% infill.



10.7.7 Custom first layers: printer speed

Placement		Setting Name			Illustration
Material	Advanced	Custom first speed	layers:	printer	

Print speed for bottom layers:

- The speed of print for the custom bottom layers.

10.7.8 Custom first layers: Repositioning speed

Repositioning speed for bottom layers:

-The speed of repositioning for the custom bottom layers.

10.7.9 Custom first layers: Over/underextrusion rate

Placement		Setting Name	Illustration
Material	Advanced	Custom first layers: Over/under-	
		extrusion rate	
			1 00% ↓ 100%

Over/under-extrusion ratio (%):

- Set extrusion ratio to a value greater than 100% to over-extrude and less to under-extrude.



10.7.10 Custom first layers: First layer thickness

Placement		Setting Name	Illustration
Material	Advanced	Custom first layers:First slice thickness	

First layer thickness (mm):

It can be useful to let this be thicker than the other layers in order to stick better to the printing surface.
Setting this value to 0 will cause ION to use the selected layer thickness in basic settings for the first layer instead.

10.7.11 Push-out speed

Placement		Setting Name	Illustration
Material	Advanced	push-out speed	

Push-out speed (mm/s):

- Push-out starts at half impulse speed and accelerates to maximum speed. By the end of push-out, the extruder speed drops to minimum speed and extrusion starts.

- Push-out happens by the end of repositioning that crosses the contour of the printed object.



10.7.12 Push-out acceleration

Placement		Setting Name	Illustration
Material	Advanced	push-out acceleration	

Push-out acceleration rate (mm/s²)

- The acceleration rate and deceleration rate of push-out.

- This is speed change per second and the value is limited by the strength of the extruder.

- Reduce this value if either the plastic gets grinded during push-out or the motor cannot keep up.

10.7.13 Push-out distance

Placement		Setting Name	Illustration
Material	Advanced	push-out distance	

Push-out distance (mm):

- Distance of material moved during retraction push-out

- A value of 0 turns off any push-out.



10.7.14 Suck-back speed

Placement		Setting Name	Illustration
Material	Advanced	Suck-back speed	

Suck-back speed (mm/s):

- Suck-back starts at half the impulse speed and accelerates to maximum speed. By the end of suckback, the extruder speed drops to minimum and extrusion ends.

- Suck-back happens just before repositioning that crosses the contour of the printed object.

10.7.15 Suck-back acceleration

Placement		Setting Name	Illustration
Material	Advanced	Suck-back acceleration	

Suck-back acceleration rate (mm/s^2) :

- The acceleration rate and deceleration rate of suck-back.

- This is speed change per second and the value is limited by strength of the extruder.

- Reduce this value if either the plastic gets grinded during suck-back or the motor cannot keep up.


10.7.16 Suck-back distance

Placement		Setting Name	Illustration
Material	Advanced	suck-back distance	

Suck-back distance (mm):

- Distance of material moved during retraction suck-back

- A value of 0 turns off any suck-back

10.7.17 Maximum Suck back during Movement

Placement		Setting Name	Illustration
Material	Advanced	Maximum Suck back during Movement	

Max suck-back during movement(s):

The suck-back can start during the last part of a path. This setting controls how long time during the end of a path, can contain suck-back.

- Put a value of 0 seconds to force the whole suck-back to be performed while waiting at the end of the path.



10.7.18 Longest repositioning line

Placement		Setting Name	Illustration
Material	Advanced	Longest Repositioning line	
			в

If a repositioning line is longer than this distance, it will do retraction even if the print head only moves inside the object.

10.8 Material settings: Expert

10.8.1 Flow factor

Placement		Setting Name	Illustration
Material	Expert	Default flow factor	
			[flow out]
			max.flow max.lineral flow
			max.flow [flow in]

Flow factor (%):

- Flow is used to compensate for material properties of different types.

- Higher flow results in increased width of the extrusion.

- The flow should be changed if you experience quality issues after switching to a different material.



10.8.2 Use advanced filament profiles

Placement		Setting Name	Illustration
Material	Expert	Use advanced filament profiles	
			[flow out]
			max.flow max.lineral flow
			max.lineral flow max. flow [flow in]

Use advanced material profile if you have more details of the material you are currently printing with. This option enables compensation for some under-extrusion by knowing when the material stops extruding linearly.

- Note that when this option is enabled, the value for "Maximum extrusion (mm^3/s) " in the Advanced tab will be the value of "Max linear flow rate (mm^3/s) ".

10.8.3 Max flowrate

Placement		Setting Name	Illustration
Material	Expert	Max flowrate	
			[flow out]
			max.lineral flow max.lineral flow max.lineral flow max.flow [flow in]

Maximum achievable flow rate (out) and the applied flow rate to achieve this limit (in).



10.8.4 Max linear flowrate

Placement		Setting Name	Illustration
Material	Expert	Max linear flowrate	[[]
			[flow out]
			max. flow max.lineral flow
			max.lineral flow max. flow [flow in]

Maximum extrusion (mm³/s):

- Defines the maximum amount of material per second that is allowed to be pushed out of the print head.

- Value is dependent on the material type, since different materials melts at different temperatures.

- This setting will limit the maximum printing speed, if necessary.

Tip: If you experience under-extrusion try to decrease this value.

10.8.5 Minimum layer speed

Placement		Setting Name	Illustration
Material	Expert	Minimum layer speed	None

When reducing the printing speed to slow down a layer, the speed can never be reduced below this value.

10.8.6 Minimum Acceleration

Placement		Setting Name	Illustration
Material	Expert	Minimum acceleration	[t]

Minimum Acceleration (mm/s²):

- Minimum acceleration is the smallest value for acceleration and deceleration rate. This is used when a



path is slowed down.

- This value is the minimum value of the Boost-slider in Printing settings -> Advanced tab. It is assigned when the Boost-slider is at 0%.

10.9 Printer settings: Others

10.9.1 Print file flavour

Placement		Setting Name	Illustration
Printer	Others	Print file flavour	None

Specify the print file flavour to fit the firmware of your printer

10.9.2 Enable pauses

Placement		Setting Name	Illustration
Printer	Others	Enable pauses	None

This setting enables pause capabilities for your printer. This includes planned pause and pause from the printer screen. Make sure that the printer has the mechanics to support this feature.



10.10 Printer settings: Workspace

10.10.1 Coordinate system

Placement		Setting Name	Illustration
Printer	Workspace	Coordinate system	

Cartesian:

This coordinate system is the regular type with a normal X, Y and Z axis.

CoreXY

Instead of carthesian coordinate system a 3D printer can use a CoreXY coordinate system. This affects the movement on the X and Y axes.

Delta

In this coordinate system, three arms move the print head around. The motors move three carriages, which are fixed to the other end of the three arms.



10.10.2 Printer front

Placement		Setting Name	Illustration
Printer	Workspace	Printer front	
			$\begin{array}{c} 4 \stackrel{z}{ \overset{z}{ \overset{y}{ y}} \\ y} \overset{y}{ \overset{y}{ \overset{y}{ y} \overset{y}{ y} \overset{y}{ y} \overset{y}{ y} \overset{y}{ y} \overset{y}{ y} y} } } } } } } } } } } } } } } }$

Choose printer front:

- To ensure the symmetry of the 3D printed object, the printers coordinate system is defined using the right hand rule. This means that X,Y and Z are always represented as shown on the picture above.

- X, Y and Z axis should therefore be plugged into the motherboard accordingly as well as the HOME define by the limit switches X1,Y1 and Z1.

- This option allows defining the front of the printer and at the same time defines where the HOME position of the printer is.

10.10.3 Print head moves on

Placement		Setting Name	Illustration
Printer	Workspace	Print head moves on	x y

Printer type:

- Specify which axis the printhead moves on.

- By default the printhead moves on X and Y, and the printbed moves on Z.



10.10.4 Workspace size: XYZ min and max

Placement		Setting Name	Illustration
Printer	Workspace	Worspace size: XYZ min and	
		max	

Workspace maximum position on X:

- Represents the last point of the printer workspace on X axis according to defined home reposition.

- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.

- Printer workspace is bounded by printer scale but should be bounded by printer bed.

Workspace minimum position on X:

- Represents the first point of the printer workspace on X axis according to defined home reposition.

- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.

- printer workspace is bounded by printer scale but should be bounded by printer bed

Workspace maximum position on Y:

- Represents the last point of the printer workspace on Y axis according to defined home reposition.

- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.

- Printer workspace is bounded by printer scale but should be bounded by printer bed.

Workspace minimum position on Y:

- Represents the first point of the printer workspace on Y axis according to defined home reposition.

- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.

- Printer workspace is bounded by printer scale but should be bounded by printer bed.



3D Labs ION User Manual

Workspace maximum position on Z:

- Represents the last point of the printer workspace on Z axis according to defined home reposition.

- Workspace minimum and maximum positions define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.

- The printer workspace is bounded by printer scale

Workspace minimum position on Z:

- Represents the first point of the printer workspace on Z axis according to defined home reposition.

- Workspace minimum and maximum size define the length of the workspace on which is possible to print. Both minimum and maximum positions are calculated according to home reposition values which defining the (0,0,0) coordinates.

- The printer workspace is bounded by printer scale.

10.10.5 End of print position XYZ

Placement		Setting Name	Illustration
Printer	Workspace	End of print position XYZ	

The "End of print" setting defines the target position in millimetres of the printhead after a successful 3D print is done.

When the print is finished, the Z-axis will move this distance away from the print head.

- Set this value to the workspace Z size to always make the bed move all the way away from the print head.



10.10.6 Heatup position

Placement		Setting Name	Illustration
Printer V	Workspace	Heatup position	
			None

Heatup position:

Represents the coordinate of the print head during the initial heating.

- The X/Y coordinates must be a position where the Z home sensor can be triggered.
- The coordinates are in mm.
- This option is made to avoid damaging the printbed due to repeated heatup at the same point.

- The heatup position can be outside the workspace so that the initial push-out does not hit the heat bed. Make sure to pick a safe position that is reachable.

10.11 Printer settings: Motion

10.11.1 Maximum acceleration

Placement		Setting Name	Illustration
Printer I	Motion	Maximum acceleration	

Maximum Acceleration Rates (mm/s²):

- Maximum allowed acceleration and deceleration for printing and repositioning. These values are limited by the strength of the motors and the mechanical friction of the axes.



10.11.2 Maximum speed limit

Placement		Setting Name	Illustration
Printer	Motion	Maximum speed limit	None

Determine the highest values allowed for the maximum speeds .

10.11.3 Minimum speed limit

Placem	nent		Setting Name	Illustration
hline	Ma-	Motors	Minimum slider speed None	
chine				

This is the target print speed when the "Speed Mode" sliders in Printing settings -> Advanced tab are moved to the left-most position (towards Quality).

10.11.4 Impulse lookahead distance

Placement		Setting Name	Illustration
Machine	Motion	Impulse lookahead distance	
			[V] 150 100 50 [t]

Impulse lookahead distance (mm):

When calculating the speed during the path, the angles between the line segments in the path decide the speed limits. If the object has a very high resolution, the angles will be very small.

This is the distance to lookahead in order to combine the angles of many small lines and give the same result for high and low resolution objects.



10.12 Printer settings: Materials

10.12.1 Nozzle diameter

Placement		Setting Name	Illustration
Printer	Materials	Nozzle diameter	

Nozzle diameter (mm):

- The diameter of the printhead's nozzle.
- This value should only be changed when the nozzle size of the printhead change.

Tip: The following settings will change automatically when Nozzle diameter change:

- Layer resolution (Layer thickness)
- Contour width
- Contour width smallest detectable
- Contour width smallest printable
- Infill line width
- Infill height (affects the maximum allowed value)

10.12.2 Enable primary heater

Placement		Setting Name	Illustration
Printer	Materials	Enable primary heater	None

Defines whether or not a heater is present for the primary print head.

When enabled, the settings related to the primary heater are enabled and heat-up G-Code is generated. When disabled, the settings related to all heaters are disabled and no heat-up GCode is generated.



10.12.3 Second printhead

Placement		Setting Name	Illustration
Printer	Materials	Second printhead	None

Used if there is no second printhead.

10.12.4 Generate tool change movements

Placement Setting Name		Illustration	
Printer	Materials	Generate tool change move-	None
		ments	

Generate tool change movement automatically during slicing. The movement will be added to the print file. Use this feature if your printer does not handle tool change in the firmware.

10.12.5 Extruder 2 offset XYZ

Placement		Setting Name	Illustration
Printer	Materials	Extruder 2 offset XYZ	None

Distance in millimeter between the nozzles of print head 1 and the nozzle of print head 2.



10.12.6 Filament diameter

Placement		Setting Name	Illustration
Printer	Materials	Filament diameter	

Diameter of the filament.

10.13 Printer settings: Bed leveling

10.13.1 Bed leveling method

Placement		Setting Name	Illustration
Printer	Bed leveling	Bed leveling method	None

Bed levelling method:

Chose if the printer is set up for manual or automatic bed levelling.

10.13.2 Z-Offset after leveling

Placement		Setting Name	Illustration
Printer	Bed leveling	Z-Offset after leveling	None

Levelling sensor offset:

The offset between the levelling sensor and the nozzle



10.14 Printer settings - Custom G-Code

Placement		Setting Name	Illustration
Printer	Custom G-Code	Explain the different fields	None

10.15 Application

Settings related to the ION application.

10.15.1 Enable REALview

Placement	Setting Name	Illustration
Manufacturer Others	Enable REALview	None

Enable the use of REALview. This feature will allow you to see 3D models next to you in augmented reality. Requires a webcam.

10.15.2 Show slider values

Placement	Setting Name	Illustration
Manufacturer Motors	Show slider values	None

If checked, the values next to the speed sliders in Printing settings -> Advanced tab will show the actual calculated speeds and accelerations. If unchecked, it will be relative values in %.



10.15.3 3D model color in workspace

Placement	Setting Name	Illustration
Advanced	3D model color in workspace	None

Print bed model color:

- Defines the color of the printing bed model.

Here you can specify the folder you wish to use for downloading models from MyMiniFactory. If you do not specify the path here, you will be asked to do so the next time you download a model.

10.15.4 Model simplification

Placement		Setting Name	Illustration
Expert	Application	Model simplification	

Disable simplification:

To decrease slicing time 3D models are simplified when opened.

Disable simplification if you experience that you have lost details on your object after slicing. Note that this will increase the slicing time significantly.

10.15.5 Desired number of polygons

Placement		Setting Name	Illustration
Expert	Application	Desired number of polygons	None

If simplification is enabled, ION will attempt to simplify the 3D model by reducing its number of polygons to the desired number of.

Higher values yield higher 3D model resolution at the cost of computation time, where as lower values



yields shorter computation time at the cost of detail.

If simplification aggressiveness is too low, the desired number of polygons might not be reached.

10.15.6 Simplification agressiveness

Placement		Setting Name	Illustration
Expert	Application	simplification aggressiveness	None

High aggressiveness causes the simplification algorithm to use all means necessary in order to reach the desired target polygon count even if it means removing large parts of the 3D model.

Low aggressiveness will result in the algorithm being more cautious with removing detail, even if it means that the desired number of polygons might not be reached.

10.15.7 Use path simplification

Placement		Setting Name	Illustration
Expert	Application	Use path simplification	✓

Path simplification reduces the number of points in the printing path. This speeds up slicing time and helps avoiding computational bottlenecks for printer models with simple controllers, at the cost of potentially losing small details.

10.15.8 Line segment resolution

Placement		Setting Name	Illustration
Expert	Application	Line segment resolution	



Line segment resolution (mm):

- Defines the allowed detail loss for path simplification
- Large values reduce path points. Too large values removes too much detail.

- Small values preserve the print details. Too small values results in high slicing time and increases the risk of computational bottlenecks for simple printer controllers.

10.15.9 Show print bed

Placement		Setting Name	Illustration
Expert	Application	Show print bed	_

Enable/Disable the visualization of the print bed in the workspace.

10.15.10 Print bed model look XYZ

Placement	Setting Name	Illustration
Manufacturer Workspace	Print bed model look XYZ	

The grid home offset setting defines the position of the 0,0,0 point of the grid relative to the printing bed model's origin.



10.15.11 Bed rotation

Placement	Setting Name	Illustration
Manufacturer Workspace	Bed rotation	

Use this setting to rotate your bed clockwise in steps of 90 degrees.

10.15.12 Color

Placement	Setting Name	Illustration
Manufacturer Workspace	Color	None

Print bed model color:

- Defines the color of the printing bed model.

10.15.13 Opacity

Placement	Setting Name	Illustration
Manufacturer Workspace	Opacity	None

Print bed model opacity:

- Defines the opacity of the printing bed model. The lower the value the more transparant it will be.



10.15.14 Model download path

Placement	Setting Name	Illustration
Advanced	Model download path	None

10.15.15 Print logging

Placement	Setting Name	Illustration
Advanced	Print logging	None

Check this if you wish to log all prints from this computer in the appdata folder.

10.15.16 REALview

Placement	Setting Name	Illustration
Advanced	REALview	None

Enable the use of REALview. This feature will allow you to see 3D models next to you in augmented reality. Requires a webcam.

