3D Printing

Manual Guide

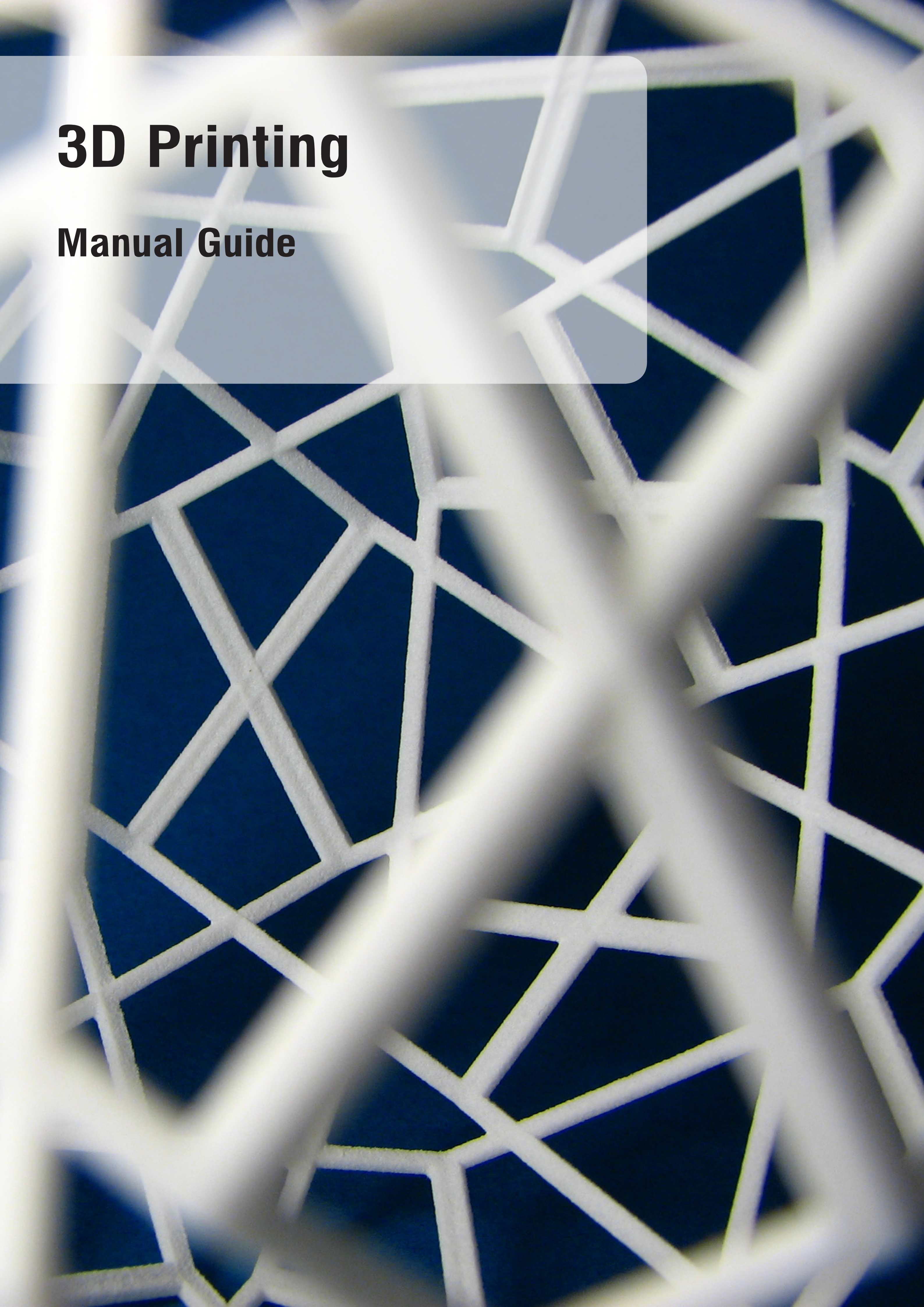


TABLE OF CONTENT

INTRODUCTION

SLA

FDM

Appendix

Additive manufacturing

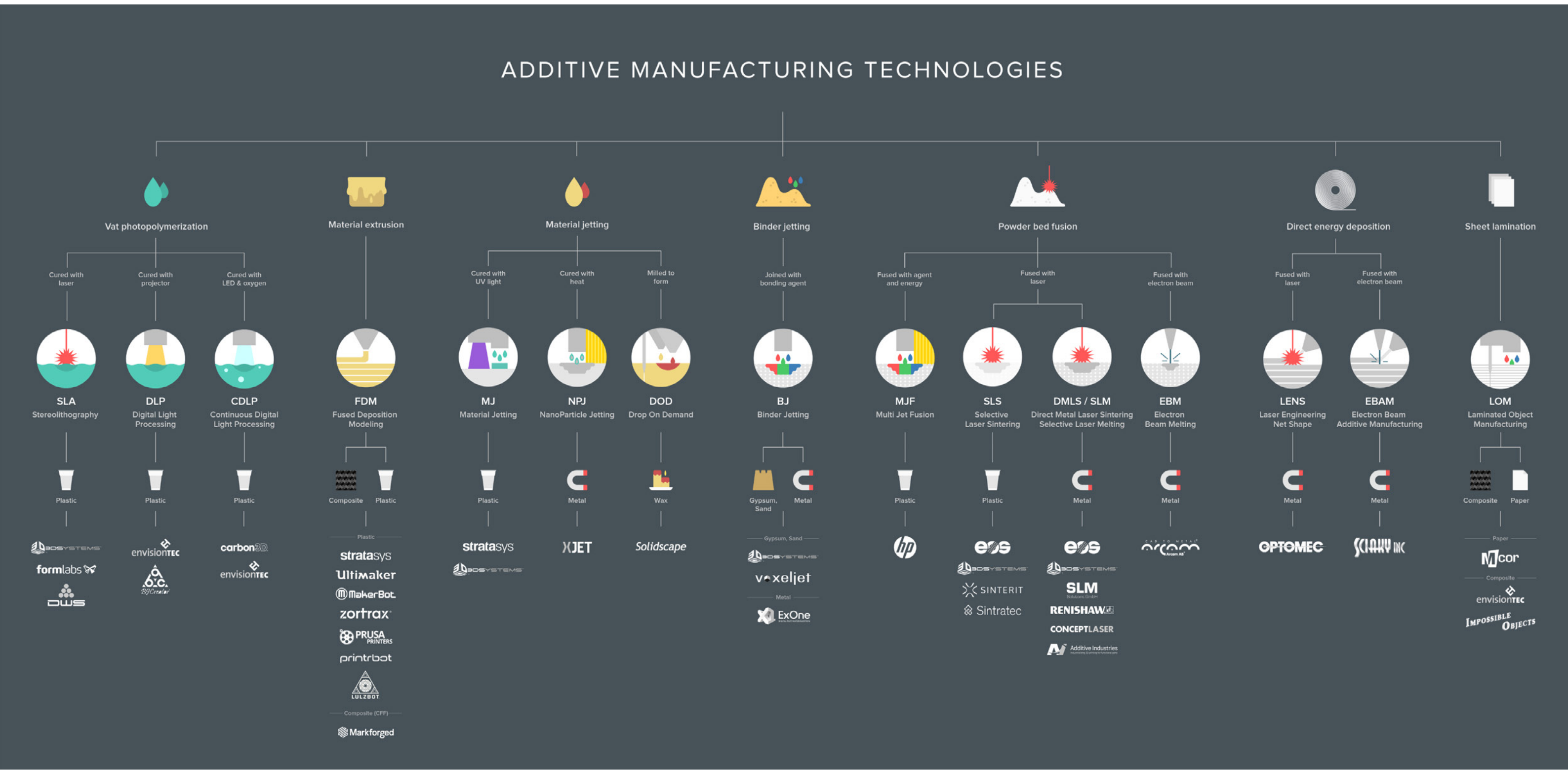
Additive manufacturing (AM) also referred to as rapid prototyping is a description for multiple techniques which are mainly focused on fast manufacturing of prototypes. Additive manufacturing is based on a CAD-CAM process. The product is designed as a virtual 3D-model (CAD). This model is then converted to a file type the machine can work with, by using a slicer ore code-generator (CAM). The machine then will create the fiscal model according to the 3D-model.

The two main streams of Additive manufacturing are CNC-processing and 3D-printing. Which can also be divided in to sub-categories.

CNC-processing
- CNC-milling
- CNC-turning

3D-printing
- Vat photo-polymerization
- Material extrusion
- Material jetting
- Binder jetting
- Powder bed fusion
- Direct energy deposition
- Sheet lamination

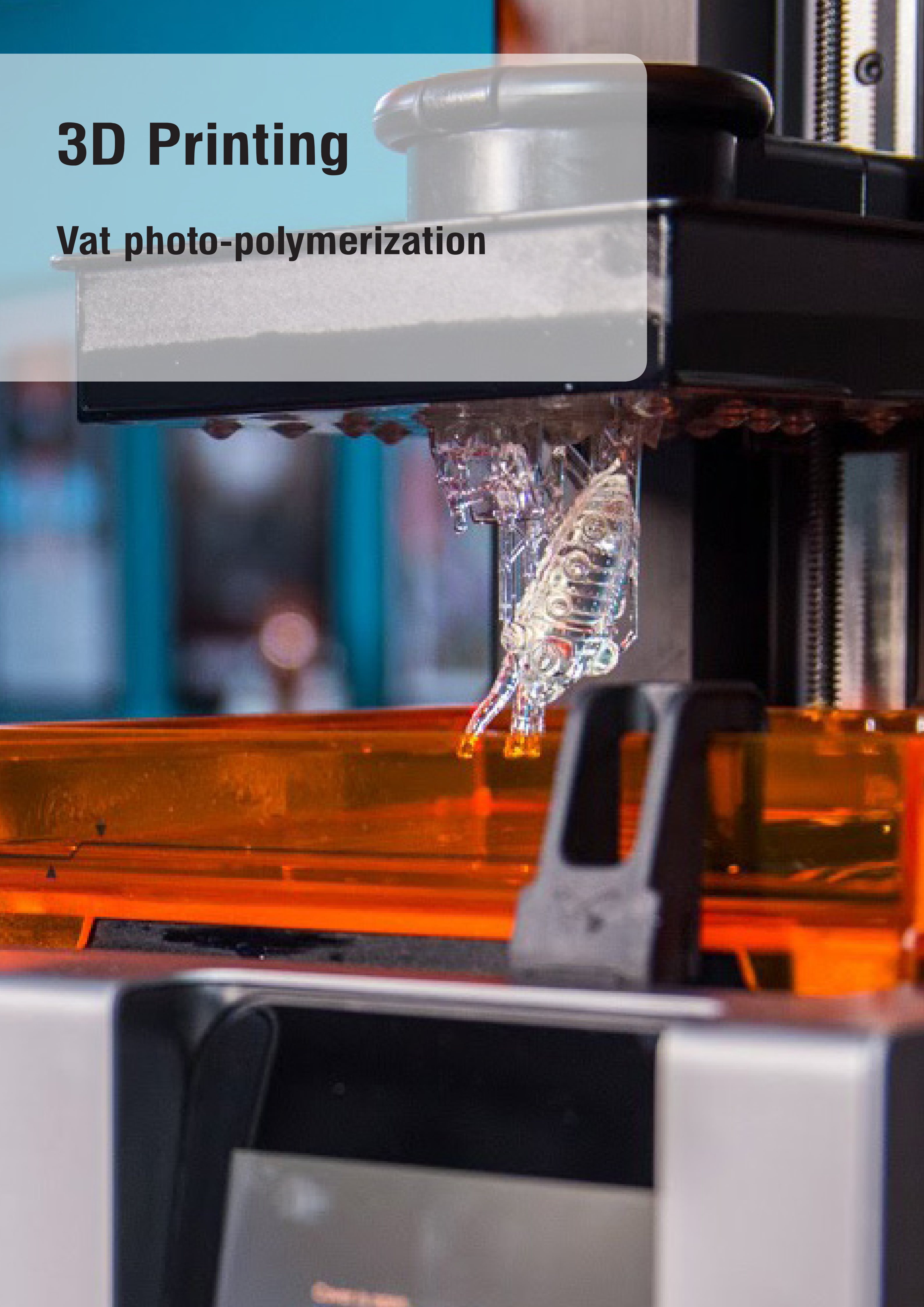
This manual will only focus on 3D-printing.



Appendix 1

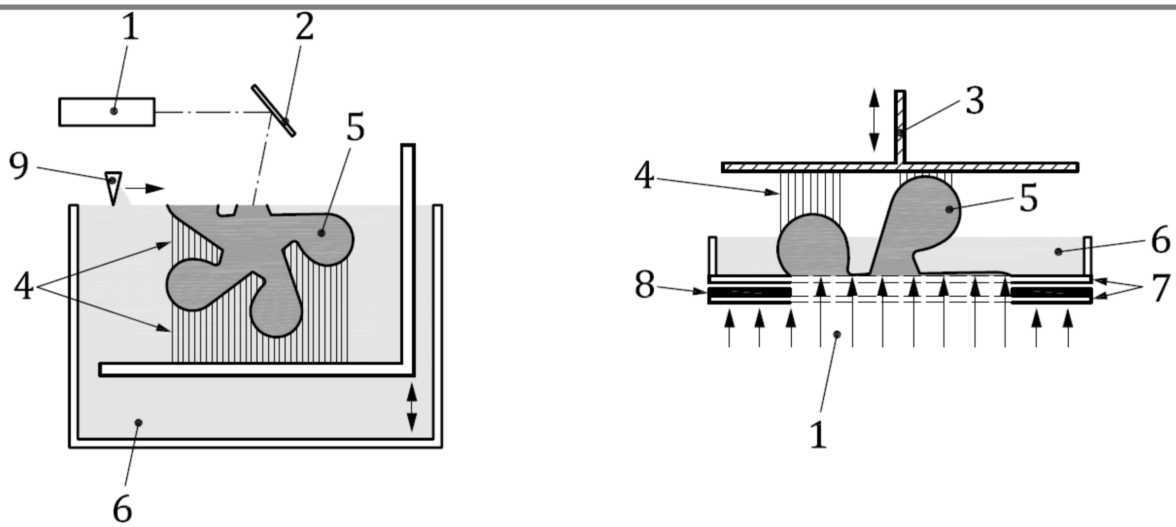
3D Printing

Vat photo-polymerization



Vat photo-polymerization

The definition of Vat photo-polymerization according to ISO 17296-1: Additive manufacturing process in which liqued photopolymer in a vat is selectivly cured by light-activated polymerization see figure 1.



a) Vat photopolymerization by laser light source

b) Vat photopolymerization by controlled area light source

Key

1 energy light source
2 tilted mirror with focus
3 build platform and elevator
4 support structure
5 product
6 vat filled with liquid photocurable resin
7 transparent plates
8 photo mask
9 recoating and surface levelling mechanism

Figure 1

Vat photo-polymerization family

Vat photo-polymerization can be divided in to three techniques
- Stereo lithography (SLA)
- Digital light processing (DLP)
- Continuous digital light processing (CDLP)

SLA, DLP and CDLP have many common characteristics for simplicity these technologies are treated as equals. Also the printer in the RPLab is a SLA printer.

Print process

Stereolithography (SLA) create 3D printed objects from a liquid (photopolymer) resin by using a UV-light source to solidify the liquid material (resin).

To create a 3D printed object, a build platform is submerged into a translucent tank filled with photosensitive thermoset polymers resin. Once the build platform is submerged, a UV-light located inside the machine maps each layer of the object through the bottom of the tank, thus solidifying the material. After the layer has been mapped and solidified by the light source, the platform lifts up and lets the swipe blade re-coat the surface with a new layer of resin. This process is repeated layer by layer until the desired object has been completed.

There are two common methods differentiated by the light source: SLA uses a laser, whereas DLP uses a projector.

SLA Prints are isotropic resulting in near identical physical properties in the x, y and z direction. There is no noticable impact on the final material properties when a part is printed parrallel ore perpendicular.

SLA 3D printers produce highly accurate parts with smooth surface finishes and are commonly used for highly detailed sculptures, jewelry molds, and prototypes. Because of their relatively small size, they are not recommended for printing large objects.

Pro:	**Con:**
High detail	Brittle
High accuracy	Support needed
Smooth surface	
Thermoset material	

Print orientation

The orientation of the part determines the location and the amount of support. It is recommended that the part is oriented so that the visually critical surfaces do not come in contact with the support structures. When Orienting a part for SLA, the biggest concern is to minimize the cross section area of each layer. The forces applied to the part during the peeling step may cause the part to detach from the build platform. These forces are proportional to the cross-sectional area of each layer. For this reason, parts are oriented in an angle.

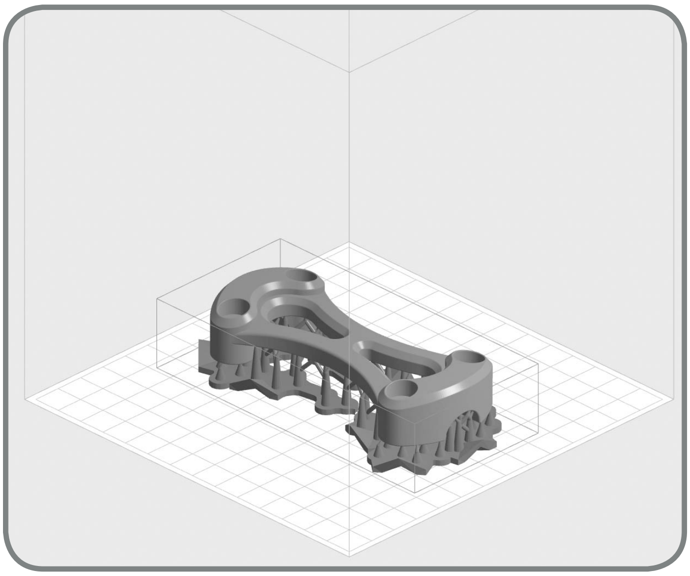


Figure 2, Horizontal orientation:
Large cross section area, high potential for print failure.

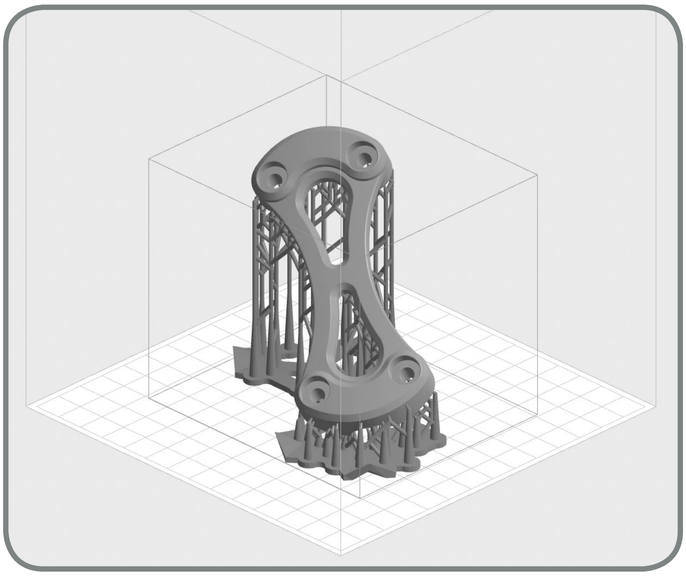
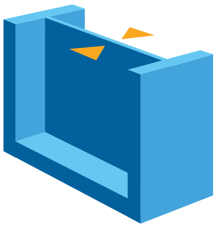


Figure 3, angled orientation:
Minimized cross section area, less potential for print failure.

SLA Printing design guide

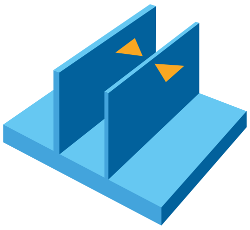


MINIMUM SUPPORTED WALL THICKNESS

Recommended: 0.4 mm

A supported wall is one that is connected to other walls on two or more sides. A supported wall smaller than 0.4 mm may warp during the peel process.

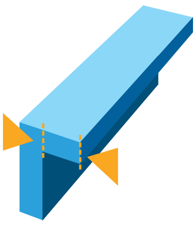
> **Note: Washing Thin Walls**
>
> Care should be taken when washing thin walls, as they may absorb IPA and swell during the cleaning process, leading to deformation of the part. Minimizing time immersed in IPA will limit this effect.



MINIMUM UNSUPPORTED WALL THICKNESS

Recommended: 0.6 mm

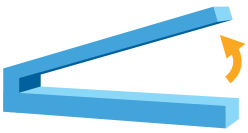
An unsupported wall is one that is connected to other walls on fewer than two sides. An unsupported wall that is smaller than 0.6 mm may warp or detach from the model during printing.



MAXIMUM UNSUPPORTED OVERHANG LENGTH

Recommended: 1.0 mm

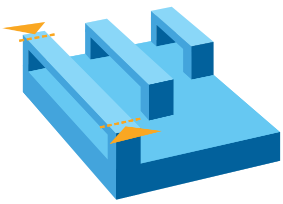
An overhang refers to a part of the model that sticks out horizontally parallel to the build platform. Printing such features without supports is discouraged, as the layers cannot maintain their structure. Horizontal overhangs will be slightly deformed beyond 1 mm and become increasingly deformed as the length of the overhang increases. You can turn on “internal supports” in PreForm to ensure your overhangs are supported.



MINIMUM UNSUPPORTED OVERHANG ANGLE

Recommended: 19° from level (35 mm long × 10 mm wide × 3 mm thick)

The overhang angle refers to the angle from horizontal that the overhang sticks out. Printing at an angle less than 19° could cause the overhang to break off the model during the peel process. Rotate your part so flat surfaces can be held up by supports if they are not already self-supporting. See the “Print Flat Surfaces at an Angle” section of our Model Orientation article.

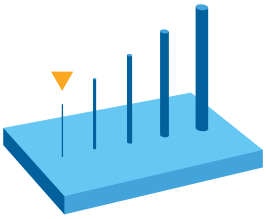


MAXIMUM HORIZONTAL SUPPORT SPAN/BRIDGE

Recommended: 21 mm (5 mm width × 3 mm thick)

A span is the distance between two intermediate supports of a structure. While printing horizontal spans is discouraged, certain geometries print well. For a 5 mm wide and 3 mm thick beam, spans longer than 21 mm are likely to fail. Wider beams must be kept shorter to avoid breaking during the peeling process.

SLA

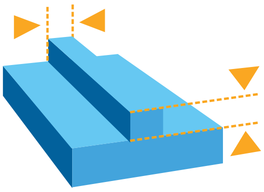


MINIMUM VERTICAL-WIRE DIAMETER

Recommended: 0.3 mm (7 mm tall) **to 1.5 mm** (30 mm tall)

A wire is a feature whose length is greater than two times its width. The ratio is key to printing wires; at 0.3 mm thickness you can print up to 7 mm tall before you start to see waving. 1.5 mm wires can get up to 30 mm tall without defects.

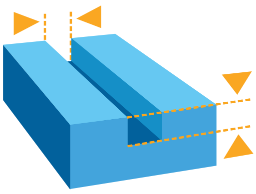
> **Note: Washing Small Wires**
>
> Similar to washing thin walls, extra care should be taken when washing small wires, as they are weakened by IPA and can easily be damaged. Minimizing the time the part is immersed in IPA will limit this effect.



MINIMUM EMBOSSED DETAIL

Recommended: 0.1 mm

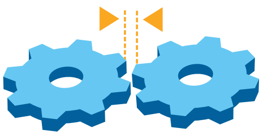
Embossed details are shallow raised features on your model, such as text. Details smaller than 0.1 mm in thickness and in height may not be visible on your print.



MINIMUM ENGRAVED DETAIL

Recommended: 0.4 mm

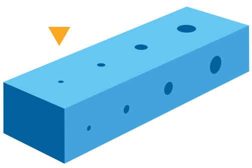
Engraved details are imprinted or recessed features on your model. Details recessed less than 0.4 mm in thickness and in height may not be visible because they will be fused with the rest of the model during the print process.



MINIMUM CLEARANCE

Recommended: 0.5 mm

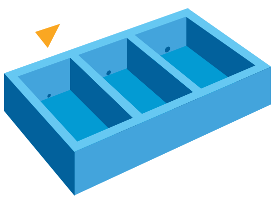
Clearance is the amount of distance needed between two moving parts of a model (e.g., the distance between gears or joints). A clearance of less than 0.5 mm may cause parts to fuse.



MINIMUM HOLE DIAMETER

Recommended: 0.5 mm

Holes with a diameter less than 0.5 mm in the x, y, and z axes may close off during printing.



MINIMUM DRAIN HOLE DIAMETER

Recommended: 3.5 mm diameter

Drain holes are recommended for resin to escape in models that are a fully enclosed cavity (like a hollow sphere or hollow cylinder printed directly on the build platform). Without drain holes of at least 3.5 mm in diameter, the part may trap resin and lead to an explosion of the print. Although drain holes are recommended, they are not entirely necessary when using a Form 2 printer.

Resolution

SLA is able to achieve much higher resolutions than FDM because it uses a laser to solidify material. SLA printing resolution in the XY-direction is dependent upon the laser spot size, and can range anywhere from 30 to 140 microns. This is not an adjustable parameter of printing. The minimum feature size cannot be smaller than the laser spot size.

Resolution in the Z-direction (or vertical resolution) varies from 25 to 200 microns. Choosing vertical resolution is a trade-off between speed and quality. For a part that has few curves or fine details there will be little visual difference between a print at 25 microns versus a print at 100 microns. For most print-jobs 100 microns is sufficient.

Hollowing and Cupping

SLA machines print a solid, dense model but if the print is not intended to be a functional part hollowing the model significantly reduces the amount of material needed as well as print time. It is recommended that the walls of the hollowed print be at least 2 mm thick to reduce the risk of failure during printing.
If printing a hollow part, drainage holes must be added to prevent uncured resin from getting trapped inside the final print. This uncured resin creates pressure imbalances within the hollow chamber, and will cause what is known as “cupping”. Small failures (cracks/holes) propagate throughout the part and will cause complete failure, or part explosion, if not corrected. Drain holes should be at least 3.5 mm in diameter, and at least one hole must be included per hollow section.

Post processing

SLA parts can be finished to a very high standard using various post processing methods.

Support removal: The support structure is broken off or cut from the model leaving a bumpy surface.

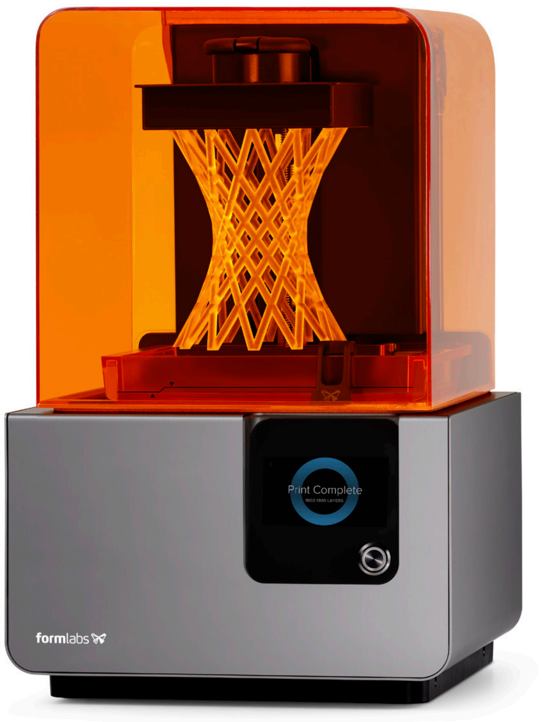
Sanding: The support bumps are sanded off leaving a flatter surface by using IPA after sanding the sanding marks will be removed.

Curing: By placing the part under an UV-light the part will be cross-linked this results in a mechanically stronger part.
Sanding:

Wed sanding: For a smoother surface finish

Spray paint: Spray paint helps to conceal layer lines and protect the material from degrading due to sunlight.

Printer specifications (Form 2)



Technique	Stereolithography (SLA)
Material	Photopolymer resin
Application	High Detailed models,
Building Platform	145*145*175 mm
Layer Thickness	25- 100 Microns
Laser Spot	140 Microns

Preform installation

To Prepare the CAD model for printing the program “Preform” is needed.

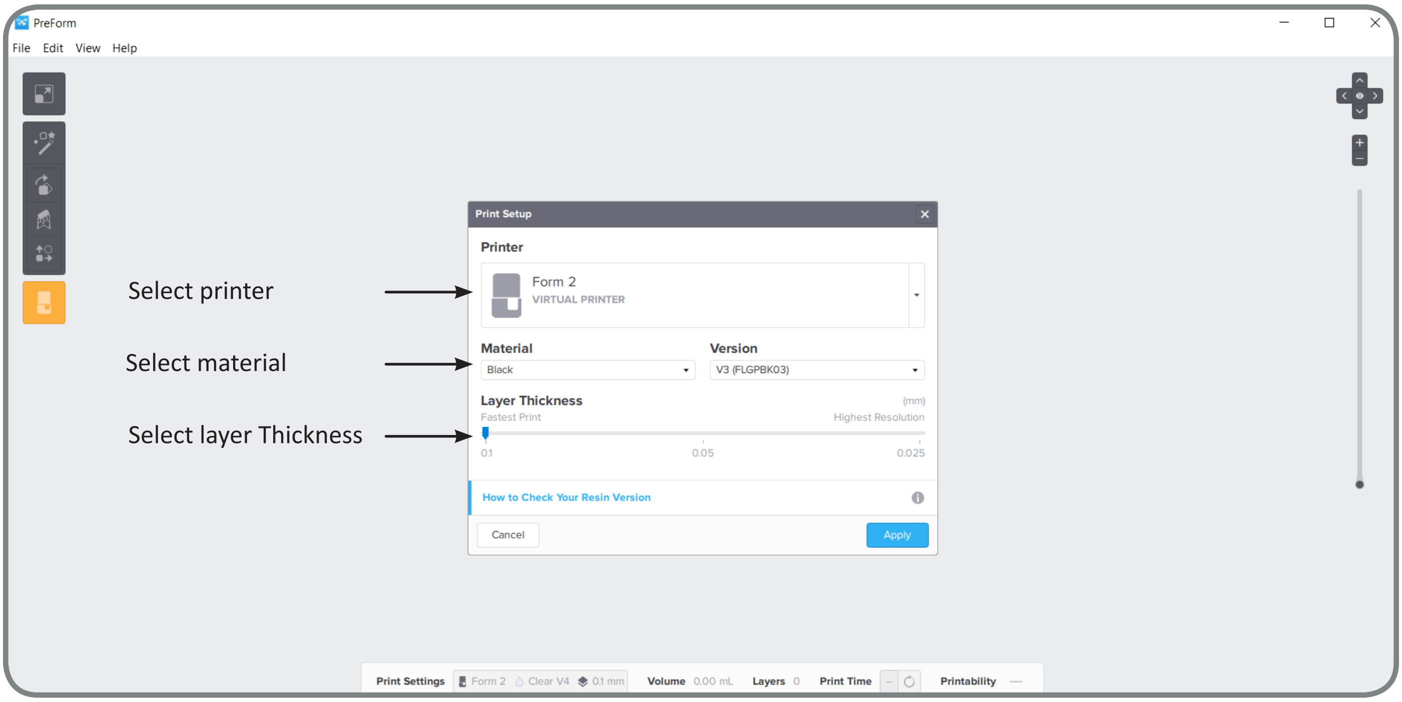
Preform installation: http://formlabs.com/tools/preform/

- Choose a installation type and follow the instructions

Cad-model preparation

First save or export the CAD file as a .STL file, This is the file type which can be opened in Preform.

- Open preform



Note:

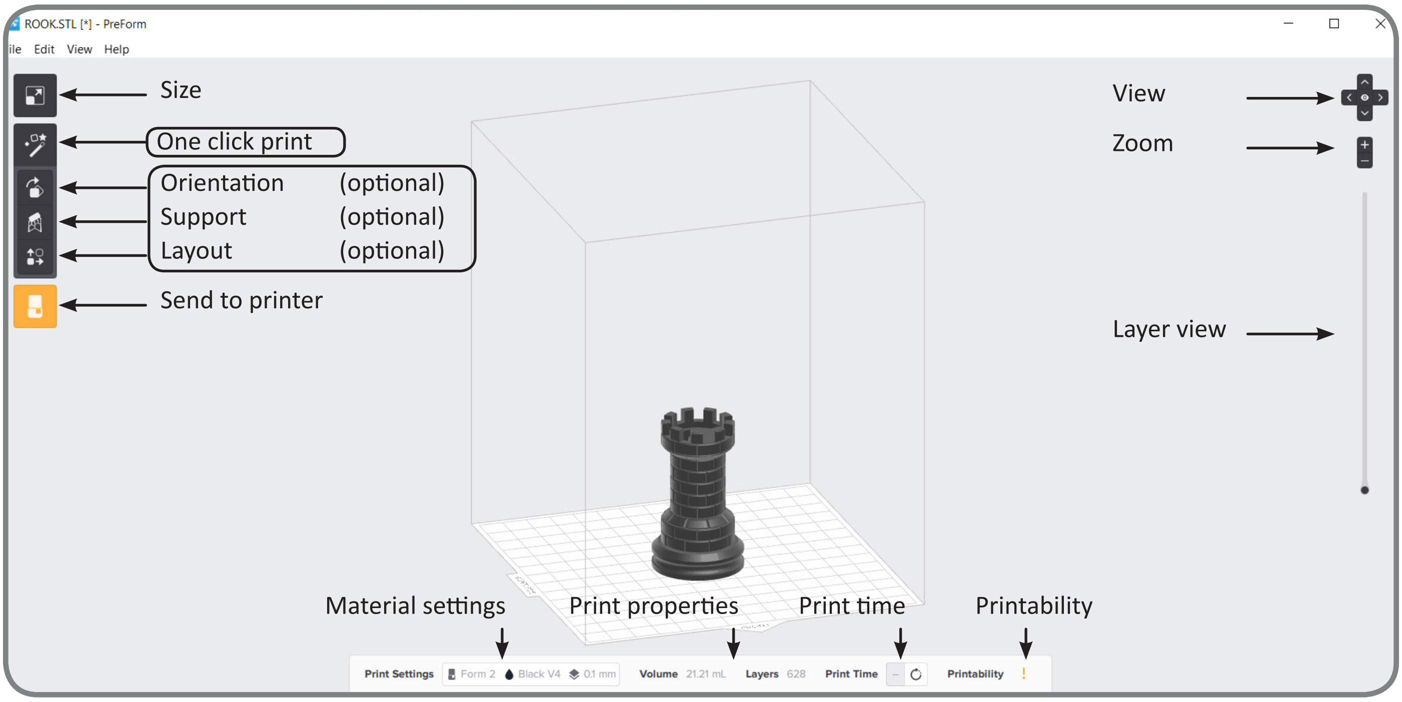
- *Printer: use “virtual printer” to set up your file*
- *Material: Available material at the RPLAB are Clear and Black the version can be changed before the start of printing*
- *Layer thickness: Standard layer thickness is 0.1 mm*

- Load the .STL file
  (By drag and dropping it in the Preform screen or by File>open (Ctrl+O)> .stl file > open

- Positioning the part
The easiest way to print your part is by hitting the “one click print” button this will calculate the orientation, generate the support and adjusts the layout for the optimal position to print. The printability should show a green check mark.

- Advanced positioning part (optional)
If “one click print” doesn’t position your part the well (support on the wrong side etc.) then the positioning, layout and support generation can be done manually by using the other buttons. Be aware of the cross section areas (minimize them)

Size: scaling the part
Orientation: Rotation of the part, selection of faces
Support: Amount and size of the support generated
Layout: Position of the part in the printer, duplication



Sending print-job to the printer
- First connect the printer to the computer via USB
- Press the orange “send to printer”button
- Select the printer
- Material in the printer should be the same as the print-job
  (if not ask RPLab personnel to change it), (if the material version is off go back and change it in the material settings see figure above)
- Press send to printer (this will take a few minutes check form2 display)

Form2
- Press print now (follow instructions on the screen)
- Press confirm
- The Form2 printer will start printing

- Fill in the spreadsheet next to the Form2

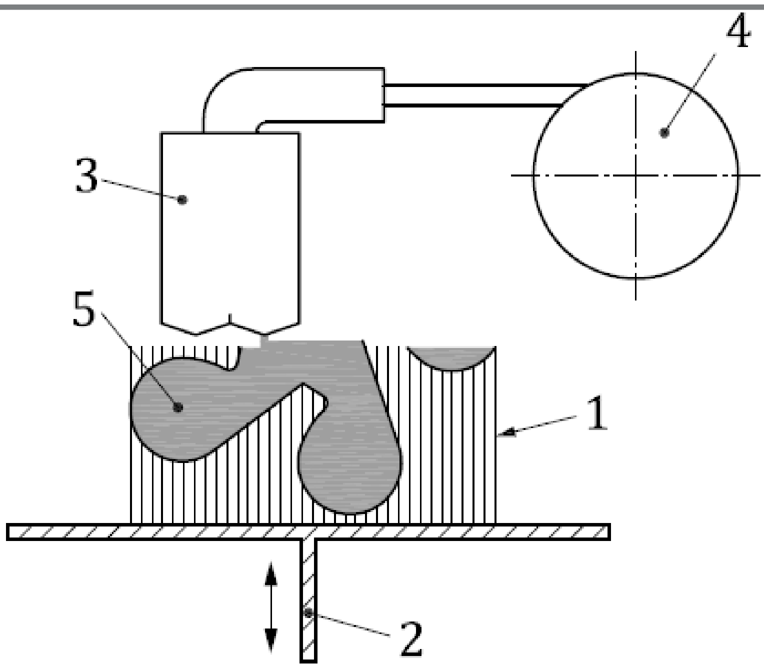
Your done, Good Job!

3D Printing

Fused Deposition Moulding

Fused Deposition Moulding

The definition of material extrusion according to ISO 17296-1: additive manufacturing process in which material is selectively dispensed through a nozzle or orifice. See Figure 1.



Key

1 support structure

2 build platform and elevator

3 heated nozzle

4 feedstock supply

5 product

Print process

Fused Deposition Modeling (FDM) creates 3D printed objects by selectively depositing melted material (thermoplastic polymer) in a pre-determined path layer-by-layer

To create a 3d printed object, the printing process starts with a string of solid material called filament. This line of filament is guided from a reel attached to the 3D printer to a heated nozzle inside of the 3D printer that melts the material. Once in a melted state, the material is extruded and selectively deposited on a predetermined path where it cools and solidifies providing the foundation for the next layer of material until the entire object is manufactured.

There are 2 FDM configurations on the market:
- The print-head moves in XY-direction and the build platform moves in Z-direction(down).
- The build platform moves in XY-direction and the print-head moves in Z-direction.

FDM is a great choice for quick and low-cost prototyping and can be used for a wide variety of applications. Due to some design and material limitations, FDM 3D printing is not recommended for more intricate designs.

Pro:	**Con:**
Low cost's	Accuracy
Fast production	Resolution
Wide range of material	Visible layers
	Anisotropic

Print orientation

The best way to print with FDM is by having a flat surface to start with, because this would not require a support layer first and helps the part to stick to the build platform, reducing the chance of build errors. Also because FDM printing is anisotropic (the strength in Z-direction is smaller than xy-direction) the function of the part and the layer orientation should be considered when printing a part (see figure 7).

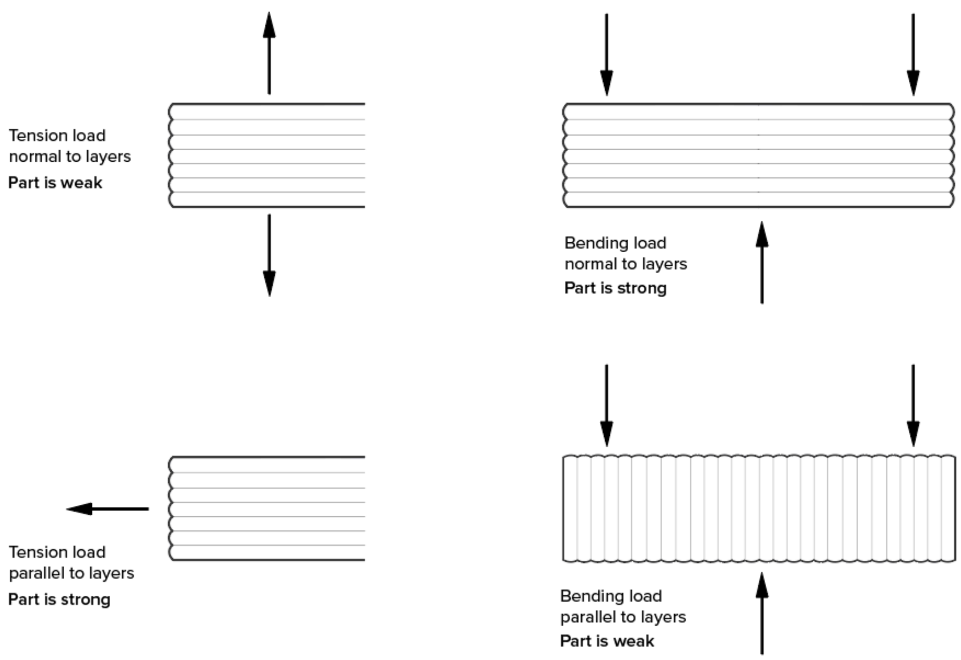


Figure 7, strong/weak Layer orientation

Warping

Warping is one of the most common defects in FDM. When the extruded material cools during solidification, its dimensions decrease. Because the sections of the part decrease at different speeds there is a build up of internal stresses that pull up the underlying layer causing it to warp. See figure 8

Reducing the chance of warping;
- Large flat areas (think of a rectangular box) are more prone to warping and should be avoided when possible.
- Thin protruding features (think of the prongs of a fork) are also prone to warping. In this case, warping can be avoided by adding some sacrificial material at the edge of the thin feature (for example a 200 microns thick rectangle) to increase the area that touches the build platform.
- Sharp corners are warping more often than rounded shapes, so adding fillets to your design is a good practice.
- Different materials are more susceptible to warping: ABS is generally more sensitive to warping compared to PLA, due to its higher glass transition temperature and relatively high coefficient of thermal expansion.
-Brims/support can prevent warping in the first layers giving a flat surface to build up on.

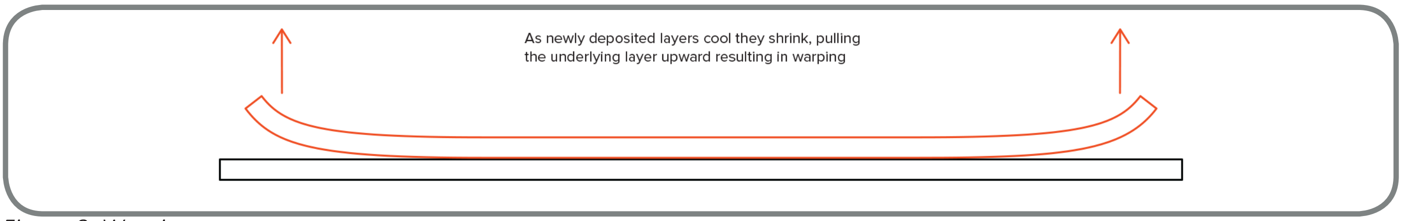


Figure 8, Warping

Support structure

Support structure is essential for creating geometries with large overhangs in FDM. The melted thermoplastic cannot be deposited on thin air, al though bridging is possible for small distances. Support is usually printed in the same material as the part and cut away afterwards. There is also support materials that dissolve in a liquid which improves the surface quality of the part.

Bridging: occurs when the printer is required to print between two supports or anchor points. Because there is no support offered for the initial layer being printed (there is nothing to build upon) and it is required to "bridge" a gap, the material will tend to sag. Bridges occur most often in horizontal axis holes found in the walls of objects or in the top layer (or roof) of hollow parts.

Overhang: Occur when the printed layer of material is only partially supported by the layer below. An overhang can usually be printed with no loss of quality up to 45 degrees, depending on the material. Above 45 degrees, support is required to ensure that the newly printed layer does not bulge down and away from the nozzle.

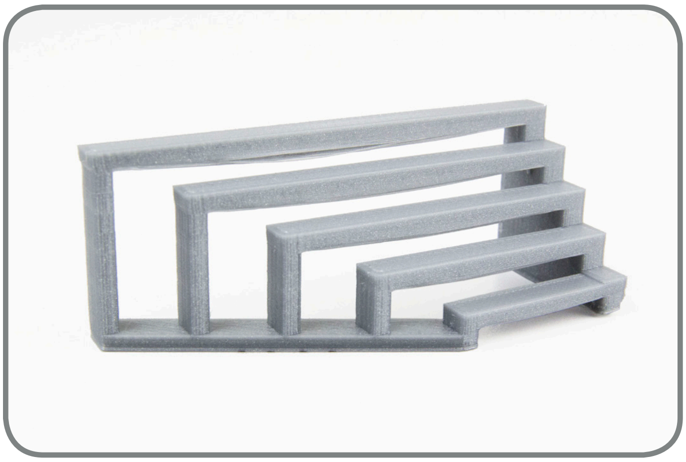


Figure 9, Bridging between two supports without support structure

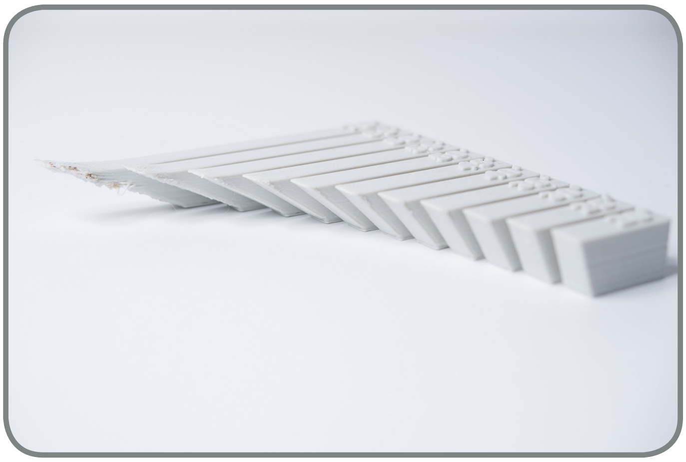


Figure 10, Overhang: Up to 45 degrees

Vertical axis holes

FDM will often print vertical axis holes undersized. The general process for printing a hole diameter and the reason the reduction in diameter occurs is:

- As the nozzle prints the perimeter of a vertical axis hole, it compresses the newly printed layer down onto the existing build layers to help improve adhesion. The compressing force from the nozzle deforms the extruded round layer shape from a circle into a wider and flatter shape (see figure 11). This increases the area of contact with the previously printed layer (improving adhesion), but also increases the width of the extruded segment. The result of this is a decrease in the diameter of the hole that is being printed.

This can be of particular issue when printing small diameter holes where the effect is greater due to the ratio of hole diameter to nozzle diameter.

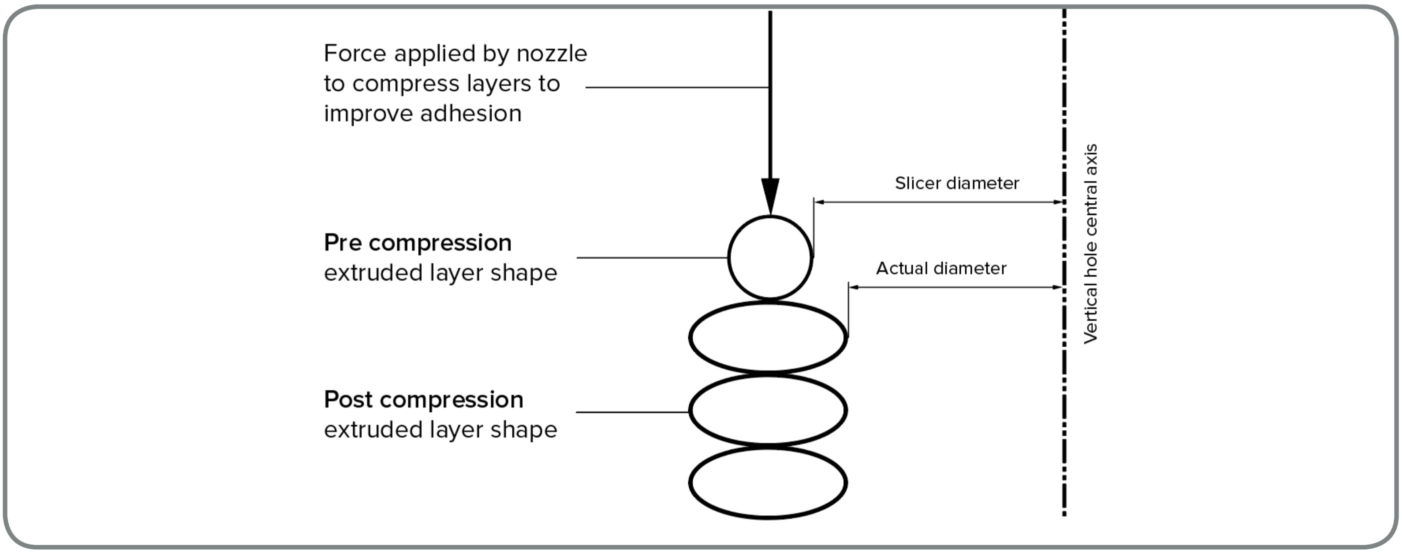


Figure 11, Layer compression

Vertical pins

Vertical pints are often printed in FDM when alignment of parts is required. It's important to understand the size which can be printed with FDM.

- Large pins (greater than 5mm) are made up with a wall (perimeter) and an infill. this gives a strong connection to the rest of the print.
- Small pins (less than 5mm) are only made up with a wall (perimeter) and no infill. This gives a weak connection ro the rest op the print. In a worst-case scenario small pins won't be printed at all because there is not enough print material to build up on.

When small pins are necessary in the part a small fillet at the base of the pin will reduce the stress concentration point. If the functionality of the small pins is critical make it a hole instead and use a off the shelf pin.

Corners

The nozzle of a FDM printer is circular, corners and edges will have a radius that is equal to the size of the nozzle. This means that these features will never be perfectly square.

- The elephants foot is the most common issue its the connection of the first layer to the print bed this layer is drawn out which can impact the ability to assemble the FDM parts. By adding a chamfer to the edges which are in contact with the build plate this problem can be overcome. If overall dimensions are critical include a 45 degree chamfer to the edges. FDM is not recommended for high precision form/fit.

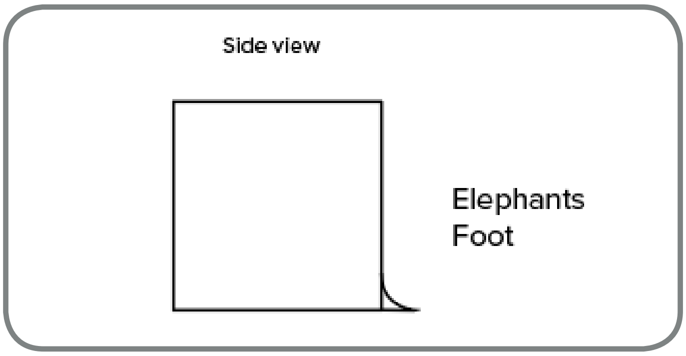


Figure 12, Elephants foot

Splitting up a model

Splitting up a model can reduce its complexity, saving on cost and time. Large amounts of support can be removed simply by splitting up a complex shape in to parts which are printed individually. The sections can be glued together once the print has been completed. If alignment is necessary a tongue and groove can be added where the part is divided.

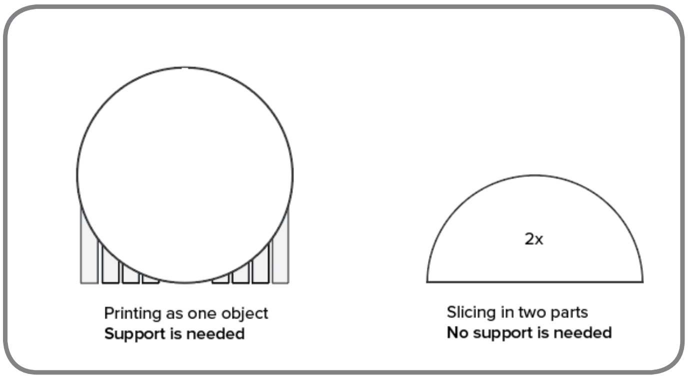


Figure 13, Splitting up a model

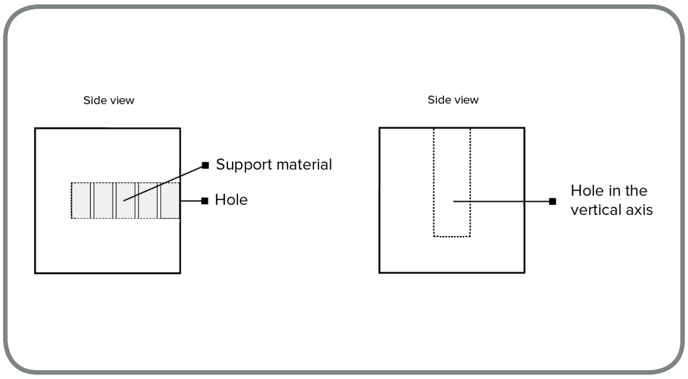


Figure 12, Hole orientation

Hole orientation

Support structures for holes is best avoided. By changing the print orientation the support structures in holes can be removed. Holes are best printed when they are in z-direction with the printer (figure 12). For prints with multiple holes in multiple directions support structure can not be avoided. Prioritize blind holes, then holes with the smallest to largest diameter, then criticality of hole size.

Material

There are a lot of materials which can be used for FDM 3D printing. The chosen material will affect the mechanical properties and the price of the printed part. The usable materials in the RPLab are listed here on the side

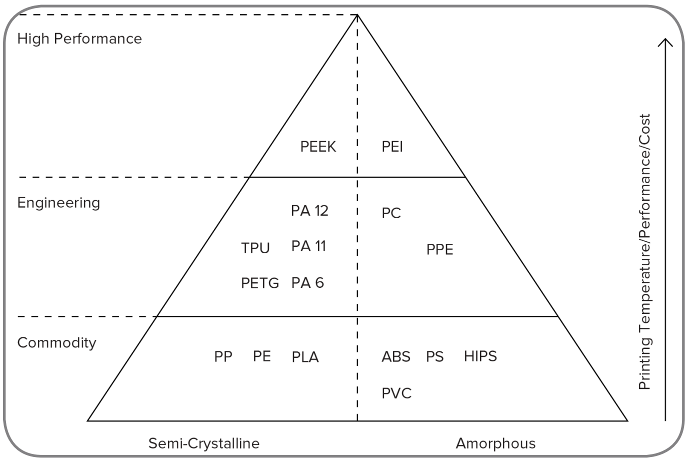


Figure 13, Thermoplastic material pyramid; The higher a material is, the better its mechanical properties.

ABS
+ Good strength
+ Good temperature resistance (85°)
− More susceptible to warping

PLA
+ Excellent visual quality
+ Easy to Print
− Low impact strength

PVA
+ Soluble in water
+ Good bonding to PLA and ABS
− Mostly used for support

Figure 14, Available materials at the RPLab

Post processing

Depending on the material FDM prints can be post processed.

Support removal: The support structure is broken off or cut from the model leaving a rough surface.

Sanding: by sanding the part a smoother surface can be accomplished also other additional surface treatments are easier to do. Sanding is not recommended for parts with 2 ore less perimeter shells.

Cold welding: a ABS split part can be welded together by using Acetone. Other materials like PLA can be glued together by using super glue (glue selection will depend on the material)

Gap filling: After sanding ore cold welding gapes can appear. Small gapes can be filled with epoxy, big gapes can be filled with autobody filler. Gapes in ABS can be filled with a ABS/Acetone slurry this will chemically react with the surface and fill any void.

Polishing: after sanding up to grit 2000 clean the print in warm water. When its fully dry polish the print with a buffing wheel ore a microfibre cloth. Giving it a injection mould look.

Priming/painting: When a part is sanded up to grit 600 it can be prime coated in two layers with a aerosol primer. after two layers of primer the part can be painted with a brush ore a spray paint.

Vapour smoothing: by vaporizing a solvent on the part a smooth fully merged surface can be created due to the dissolving of the outer shell.

Dipping: The printed part in submerged in a appropriate solvent for a few seconds aggressively smoothing the surface of the printed part. After drying white marks can appear on the print these can be dealt with by suspending the part above the solvent bath for some time restoring the collar and ensuring a shiny outer layer.

Printer specifications (Ultimaker 3)

Technique	Fused deposition moulding (FDM)
Material	Filament (PLA, ABS, ect.)
Application	Fast Prototyping, multiple colors
Building Platform	200*200*200 mm
Layer Thickness	60 - 200 Microns
Nozzle Diameter	Dual extruder; A-B 400 microns

FDM

Cura installation

To Prepare the CAD model for printing the program “Cura” is needed.

Cura installation:
https://ultimaker.com/en/products/ultimaker-cura-software

- Download Cura (choose;I don’t want to share any information)
- Install Cura

- First time use: select “Ultimaker 3”

Cad-model preparation

First save or export the CAD file as a .STL file, This is the file type which can be opened in Cura.

- Open Cura

- Load the .STL file
  (By drag and dropping it in the Cura screen or by the open file button > .stl file > open

Positioning the part
The part is positioned axiomatically in the middle of the build plate, when printing multiple models the layout can be changed so the parts all fit on the build plate.

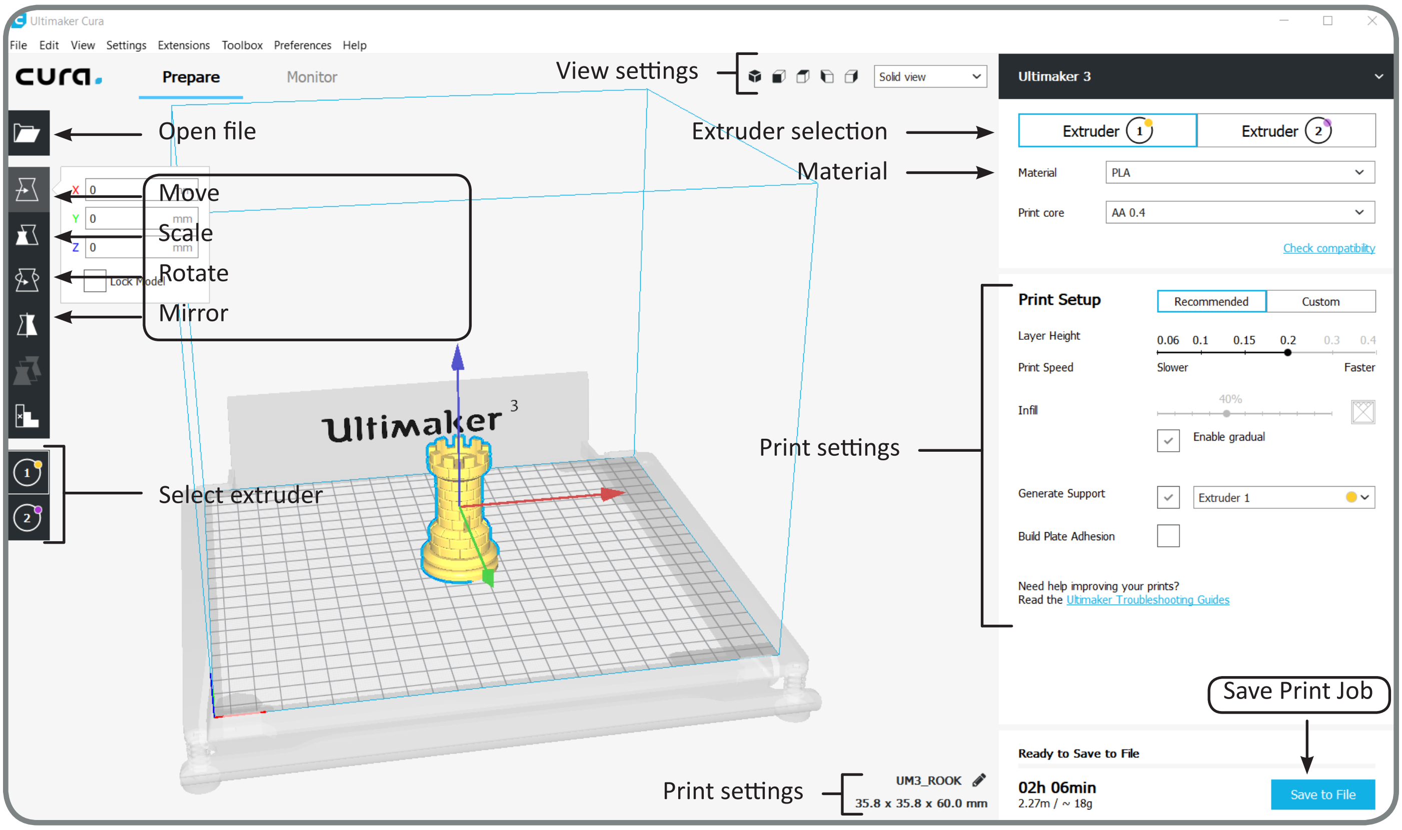
Move: Moving the part x,y,z direction (use the arrows)
Scale: Scaling the original part
Rotate: Rotating the part
Mirror: mirror the part

Print job setup
- Select the extruder which is going to print the part.
- Select material (normal extruder1; PLA extruder2: ABS)
- Select layer height (normal 0,1)
- Select the infill density (depends on the function of the part)
- Check support if necessary
- Check build plate adhesion if necessary
- Put a USB in the computer
- Save to removable disc
- Eject USB
- Put it into the printer (USB needs to be in the printer during the print job)

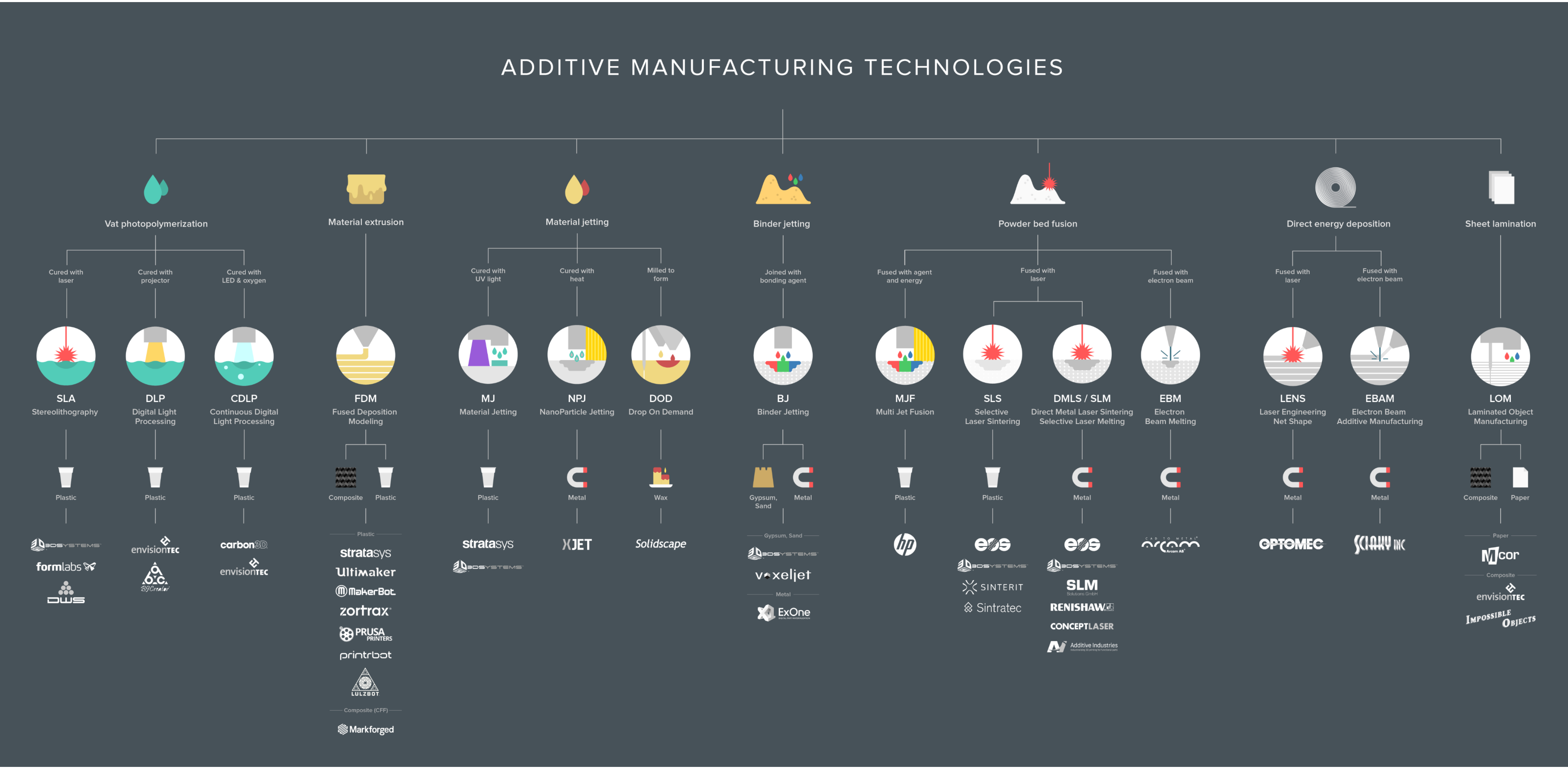
Ultimaker
- Rotate the button select Print
- Select the print job
- Hit print

- Fill in the spreadsheet next to the Ultimaker3

Your done, Good Job!



Appendix 1


ADDITIVE MANUFACTURING TECHNOLOGIES
Vat photopolymerization
Material extrusion
Material jetting
Binder jetting
Powder bed fusion
Direct energy deposition
Sheet lamination
Cured with laser
Cured with projector
Cured with LED & oxygen
Cured with UV light
Cured with heat
Milled to form
Joined with bonding agent
Fused with agent and energy
Fused with laser
Fused with electron beam
Fused with laser
Fused with electron beam
SLA
Stereolithography
DLP
Digital Light Processing
CDLP
Continuous Digital Light Processing
FDM
Fused Deposition Modeling
MJ
Material Jetting
NPJ
NanoParticle Jetting
DOD
Drop On Demand
BJ
Binder Jetting
MJF
Multi Jet Fusion
SLS
Selective Laser Sintering
DMLS / SLM
Direct Metal Laser Sintering
Selective Laser Melting
EBM
Electron Beam Melting
LENS
Laser Engineering Net Shape
EBAM
Electron Beam Additive Manufacturing
LOM
Laminated Object Manufacturing
Plastic
Plastic
Plastic
Composite
Plastic
Plastic
Metal
Wax
Gypsum, Sand
Metal
Plastic
Plastic
Metal
Metal
Metal
Metal
Composite
Paper
Plastic
Composite (CFF)
Gypsum, Sand
Metal
Paper
Composite