

WHITE

PAPER

09

**A full walkthrough of the  
3D printing workflow**

---

# Table of contents

---

01

**The 3D Printing Workflow**  
Page 3

---

02

**Modelling**  
Page 5

---

03

**Generating an STL file**  
Page 8

---

04

**Slicing**  
Page 11

---

05

**3D Printing**  
Page 12

---

06

**Post-Processing**  
Page 13

---

07

**Final test**  
Page 16

---

# 01

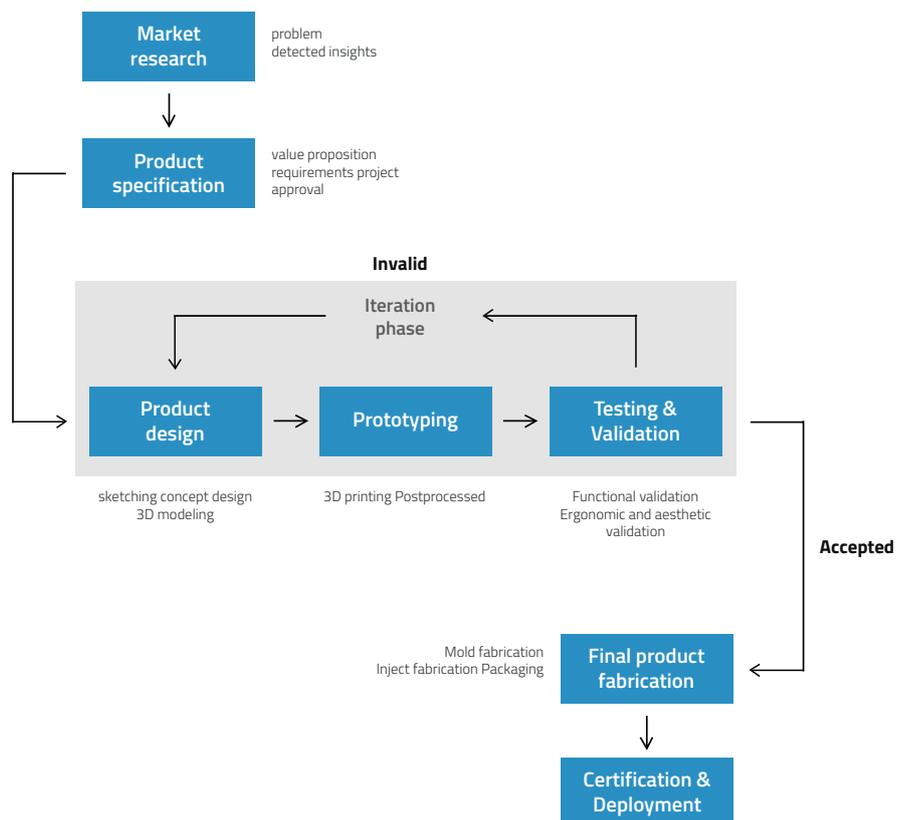
## The 3D Printing Workflow

The 3D Printing Workflow is the process comprehending all stages from the idea to the fabrication of a final part using additive manufacturing technologies. Understanding which are the necessary steps to be taken in each stage of this process is the secret to better plan for it, optimizing time and resources.

The main stages to be considered within this workflow for FFF 3D printing, in all of which we will dive deeper in this White Paper, are:

- Modelling or obtaining a pre-made model
- Generating an STL file
- Slicing
- Printing
- Post-Processing

It is important to note that this workflow is a cyclic process and can be used to refine designs. Therefore, one cycle can be used to generate a final part, but if the functional testing fails, it will be necessary to return to the first stage, and re-consider the characteristics of the model itself.

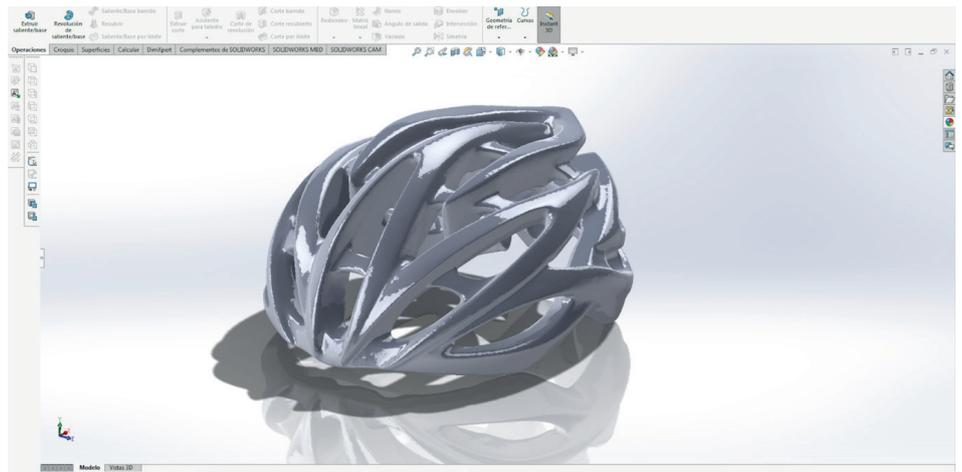


## **The 3D Printing Workflow**

It is also important to keep in mind that when the printed parts need to compile with any mechanical, thermal or chemical requirements, a material selection stage is a must. Material selection is included in every designing process of physical objects, and its main goal is to ensure that the final product requirements are met. For that reason, it is vital for the designer to know about the properties of the different 3D printing filaments that can be used, and their behaviour under working conditions.

# 02

# Modelling



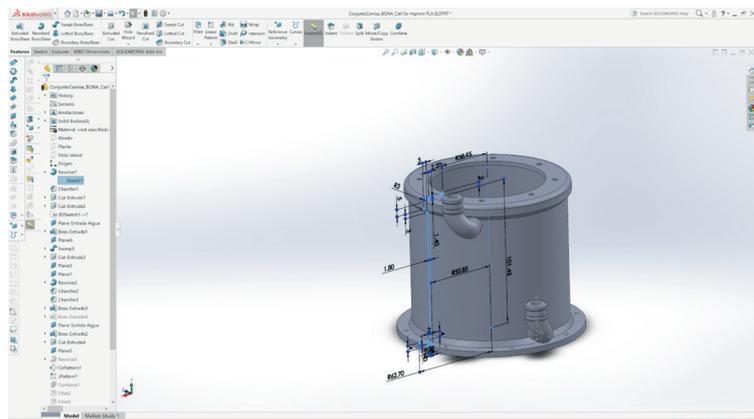
The first step in any additive manufacturing process is obtaining or producing the 3D model that is to be printed.

Modelling is the process of representing a 3D object in a mathematical way, and can be done manually, using computer-aided design (CAD) software or by 3D scanning objects to represent them digitally. Another option would be using a pre-made 3D model, which can be obtained from a variety of sources.

## CAD software

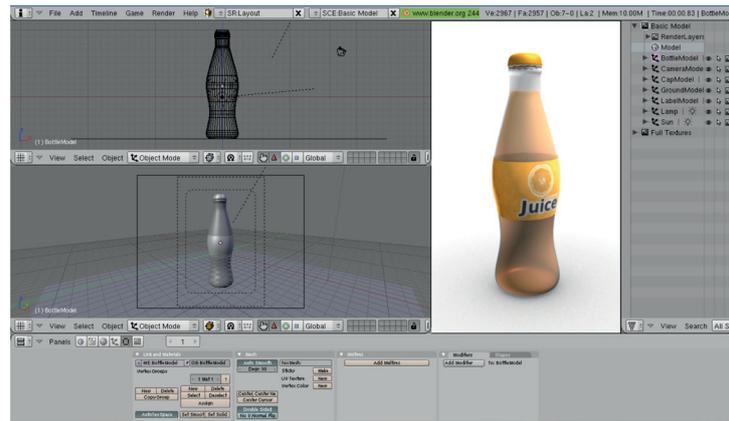
There are many CAD software programs on the market and most of them are compatible with additive manufacturing. The main two types of CAD software are those based on Mechanical and Organic design - the mechanical one is focused on dimensions, while the organic one is focused on proportions.

Mechanical design:



# Modelling

Organic design:



## 3D Scanners

As previously mentioned, the model can also be obtained through a digital scan. 3D scanning is the process of analyzing a real object to collect data on its shape and appearance.

3D scanning technologies can be classified in categories:

- Laser triangulation 3D scanning technology, which projects a laser beam on a surface and measures the deformation of the laser ray.
- Structured light 3D scanning technology measures the deformation of a light pattern on a surface to 3D scan the shape of the surface.
- Photogrammetry, which reconstructs in 3D a subject from 2D captures with computer vision and computational geometry algorithms.
- Contact-based 3D scanning technology relies on the sampling of several points on a surface, measured by the deformation of a probe.
- Laser pulse 3D scanning technology is based on the time of flight of a laser beam. The laser beam is projected on a surface and collected on a sensor. The time of travel of the laser between its emission and reception gives the surface's geometrical information.

Basically, the 3D scanner projects a laser over the object, collecting coordinates in the space that contains the object. The more points are collected, the more resolution will be obtained. The next step would be obtaining a point cloud that a software joints, forming a mesh that eventually defines the object.

# Modelling

## Pre-made 3D Models

Several online marketplaces for 3D content allow individual designers to sell or share for free the content they have created. This way, artists can distribute their designs further, and companies can save money and time by buying pre-made models instead of creating one from scratch through their own resources.

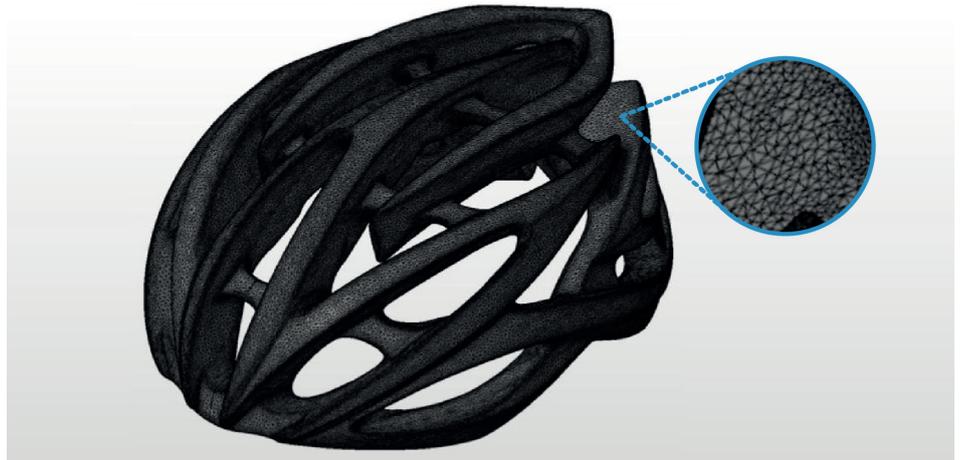
Some of the best-known 3D-model marketplaces are:

- [Thingiverse](#)
- [CGTrader](#)
- [Sketchfab](#)
- [GrabCAD](#)
- [Clara.io](#)
- [Myminifactory](#)
- [Cults3D](#)

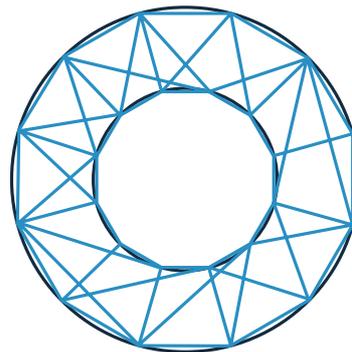
# 03

## Generating an STL file

---



Once the CAD model is ready, a vital stage during the 3D printing workflow is converting it into an STL (stereolithography) file. STL uses triangles (polygons) to describe the surfaces of an object, without any representation of colour, texture or other common CAD model attributes.

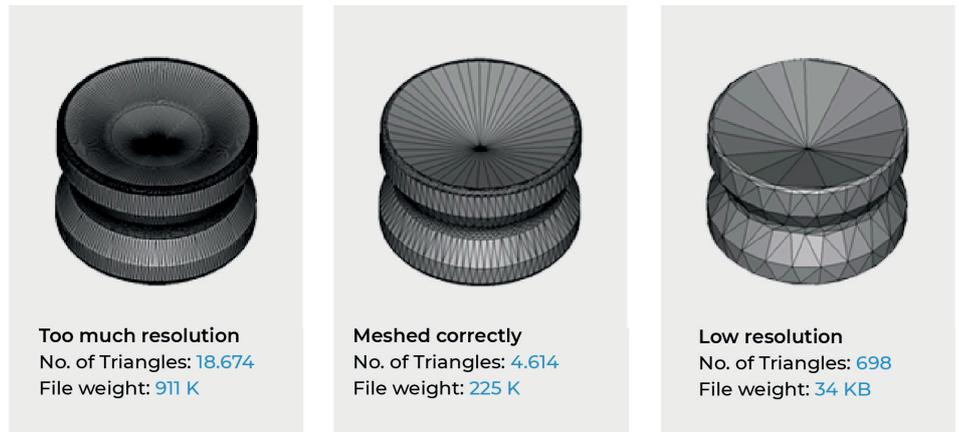


Some considerations must be taken into account when exporting the model to an STL file. As the process creates a mesh of triangles, the resolution can be controlled by changing the density of the mesh:

- If the mesh density is too low, the model will show triangles on its surface when printed. This means that the file will lose resolution, a result not necessarily undesirable, as it creates low-poly models, but that needs to be taken in consideration in order to make an informed choice.

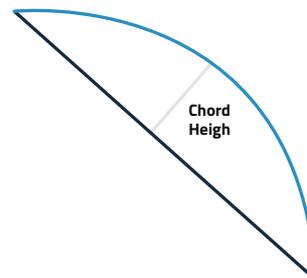
## Generating an STL file

- If the mesh density is too high, it will increase the size of the file (which will make it more difficult to handle and process) and the printer will not be able to print the finest details of the part.

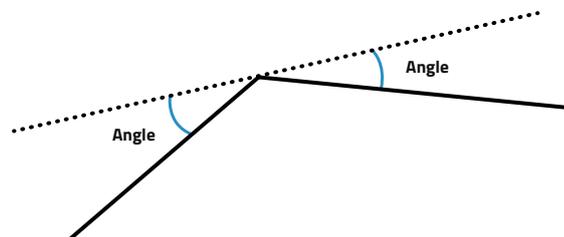


The exported resolution of the STL file can be adjusted in the CAD software. To always ensure the highest resolution no matter the difficulty of managing the file, the best option is to work with two parameters:

- **Chord Height:** The maximum distance between the surface of the CAD model and the STL file. Reducing the height will increase the resolution.



- **Angle:** The angular tolerance limits the angle between the normals of adjacent triangles.



## Generating an STL file

## Repairing an STL file

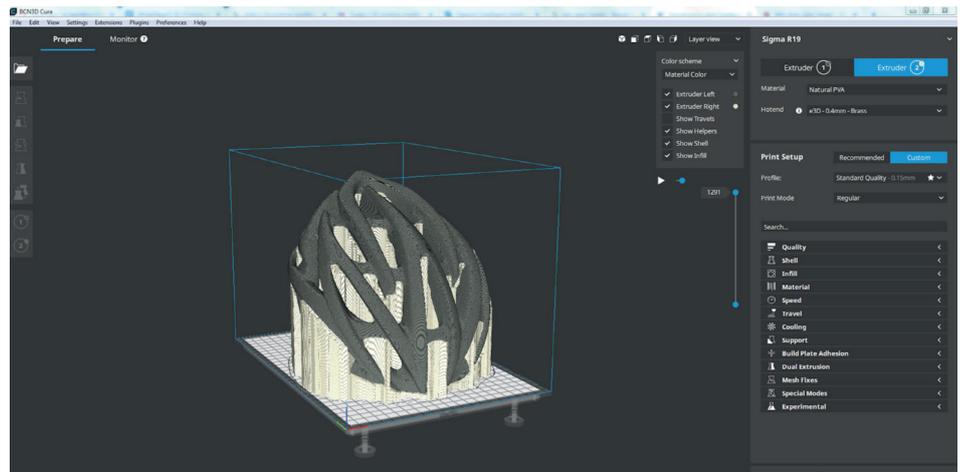
Sometimes, the STL can present problems related with the mesh. The most common issues that an STL can show are:

- Holes or gaps in the mesh: The adjacent triangles fail to share two common vertices and the missing triangle produces a hole on the model.
- Flipped normals: Each triangle have a normal vector that tells the slicer software which side of the triangle is from inside the model and what is outside. However, if some normal vector is flipped, the slicer gets confused regarding where to print.
- Intersecting and overlapping triangles: When two surfaces overlap or cross one another due to the complexity of the internal geometry.
- Bad edges: The edges of triangles are not properly connected to each other

To solve these issues, additional software such as [Magics](#) by Materialise (paid), [Netfabb](#) by Autodesk (free version), [3D Builder](#) (preinstalled on Windows), [Meshmixer](#) (free) or [Meshlab](#) (free) can be used.

# 04

## Slicing



Once the STL file has been correctly generated, it needs to be loaded into a slicing software. The slicing software is used in the additive manufacturing processes to convert a 3D model into specific instructions for the printer, by turning the STL file into a G-code with printer commands:

```
G1 X188.689 Y200.112 E0.60747
G1 X188.438 Y199.965 E0.6159
G1 X185.608 Y198.22 E0.70971
G1 X183.733 Y196.892 E0.77455
G1 X183.545 Y196.749 E0.78121
G1 X181.362 Y194.981 E0.86047
```

The slicer divides the object into horizontal layers, and for each layer describes the necessary movements to extrude the plastic. Apart from the coordinates of the extruder, the software includes information about the temperature, the speed, the material flow, supports... If these instructions are precise and have the right parameters, the prints will achieve top quality. All slicers available in the 3D Printing Market can achieve similar smooth surfaces for normal structures, but the quality of overhangs and bridges is directly affected by the slicing software used. A good slicing software should not only calculate each layer separately, but also look into the layers above and below to optimize the extrusion paths. If the printhead is accelerated in areas where an overhang or bridging is detected, the quality of the results will be much higher.

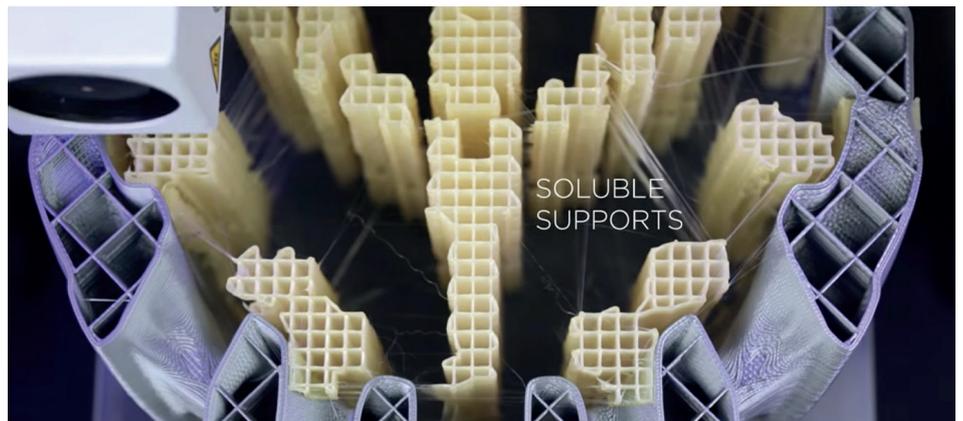
The main parameters to be modified are:

- Nozzle size
- Filament
- Printing profile

It is also important to correctly place the part in the printing surface and optimize the orientation in order to take advantage of the anisotropy or to minimize support structures.

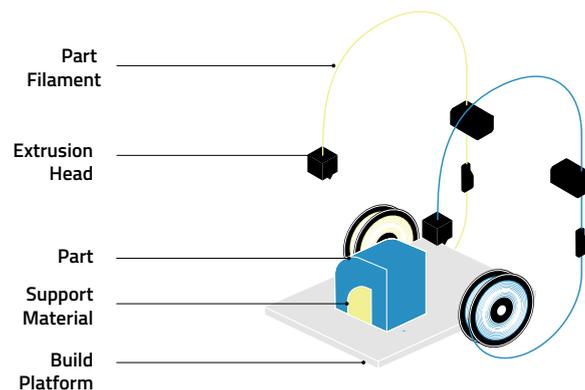
# 05

## 3D Printing



The process of FFF technology consists of building objects by [depositing melted material layer-by-layer, using thermoplastic polymers](#) as raw material.

The printer takes a filament of a thermoplastic material and pushes it to the print head using gears. The print head will melt the filament at an adequate temperature and will depose it through a nozzle along the XY-plane to create one layer. Once the layer has cooled down, the platform moves down the height of one layer and deposes the path of material to build the second layer. The process continues until the full model is completed.



In this stage, correct maintenance of the printer and accurate calibration are vital to produce high-resolution parts.

Currently, the market of 3D printers offers a wide range of capabilities and requirement ranges, and it can be at times overwhelming. [Download our ultimate guide to choose a 3D printer](#) and discover all the main features that differentiate all printers available in the market nowadays. Matching your own company's goals, needs and priorities will help you narrow the search and choose the 3D printer that better fits your corporate needs.

# 06

## Post-Processing

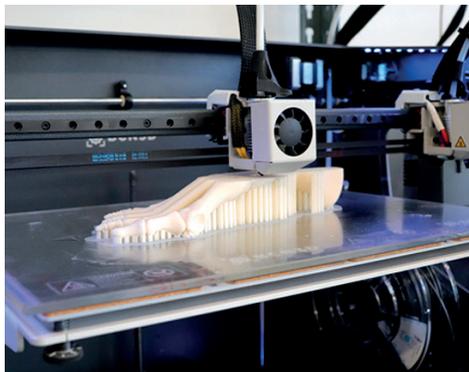
---



For most of the additive manufacturing technologies, it is required to post-process the printed parts in order to get them ready for use. However, in the case of FFF parts it is not always mandatory, and its main purpose would be to hide visible layer lines, to remove supports or to mitigate the anisotropic behaviour.

Removing supports is the most important type of post-processing of the parts that have complex geometries. There are two types of supports:

- Standard supports, which are made with materials which are not dissolvable and have to be removed mechanically. They are printed faster than the dissolvable supports but do not generate a smooth surface finish.



## Post-Processing

- Dissolvable supports are removed introducing them into a bath of solvent. The most common solvable support materials are:

- PVA
- HIPS
- BVOH



After support removal and depending on the material used for the print, there is a big range of post-processing techniques that can be applied to the part:

### - Sanding

Sanding 3D printed models can be a great way to eliminate layer lines in preparation for painting, silicone molding, or vacuum forming. This process can be done both manually, using sandpaper, or with an electric sander.

### - Cold welding

If the 3D model was larger than the build plate and needed to be split into smaller pieces, after printing it will be necessary to reattach all the pieces.

Before adhering the parts together, it's advisable to rough up the two surfaces that will stick together, giving more surface area for the adhesive to stick to and allowing for a stronger bond. This can be achieved by sanding the 3D print in those parts that need to stick until they are noticeably rougher than the clean sides.

For most materials (like PLA, ABS, or PETG), cyanoacrylate (or superglue) will be the most common and easiest way to keep two pieces together.

### - Gap filling

If there are holes on the surface of the 3D print or in-between structural components that can't be sanded or smoothed out, they will need to be filled. Bondo Body Filler, filler primers, Evercoat Body Filler, Bondo Glazing and Spot Putty, wood filler, and 3M Acry-fill Spot Putty are all excellent choices to fill in the larger gaps.

### - Polishing

Polishing a 3D model is done with the aid of a polishing compound and a buffing wheel. To do this, the 3D print needs to be dried out, to be later sprayed with a polishing agent onto its surface and smoothed using a buffing wheel or a small cloth.

### - Priming & painting

Painting 3D printed parts is a vast world of acrylics, enamels, sprays, and airbrushes. Once the materials to be used have been chosen, the most straightforward process would be to prime, dry, paint, dry, varnish, and dry again.

## Post-Processing

### - Vapor smoothing

In order to polish certain materials, such as ABS, a polishing agent is not enough. Acetone in both liquid or vapor form can be used then, always keeping in mind the necessary safety measures.

Other post-processing techniques that can be used with 3d printed parts are dipping, epoxy coating and metal plating.

# 07

## Final test

---

After all steps have been completed, it will be necessary to test the part. Depending on the final application of the 3D printed piece, the functionality, the mechanical and/or chemical performance, the aesthetic or all of these characteristics will have to be validated through specific tests.

Implementing this workflow will help you with introducing FFF 3D printing in your business's day-to-day processes in the simplest way, as it contains all the steps needed to 3D print a model, achieving the best possible results along the way.

---

# Maximizing productivity with the most versatile 3D printers

If you are considering implementing 3D printing technologies into your workflow, ask for a BCN3D Sample Part in the following link in order to see the results of a professional FFF 3D Printer [Request a Free sample >](#)

Explore more about 3D printing. [Learn more >](#)

Wondering what's new in the 3D printing world? [Use Cases >](#)

Request a quote for a professional desktop 3D printer. [Request a quote >](#)

