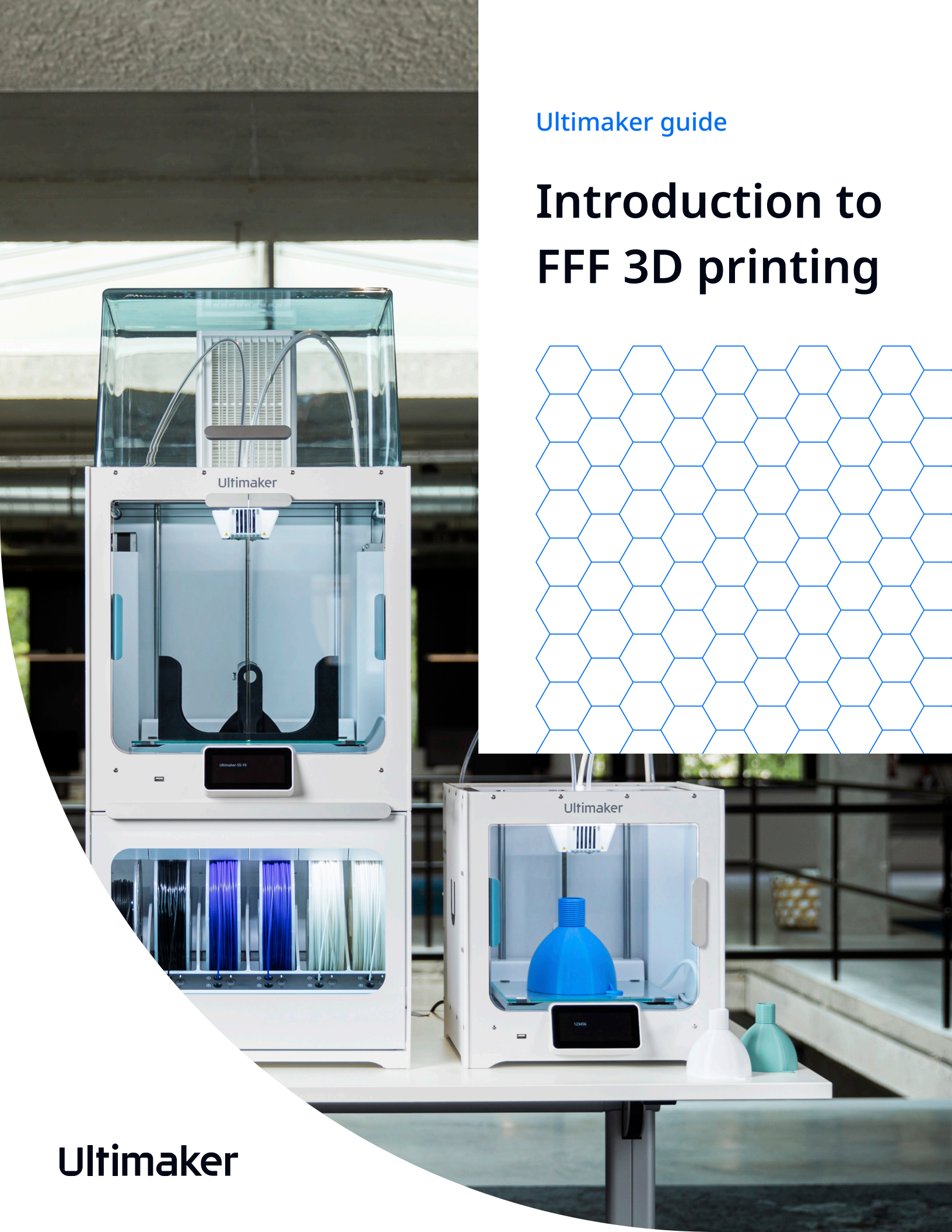
Ultimaker guide

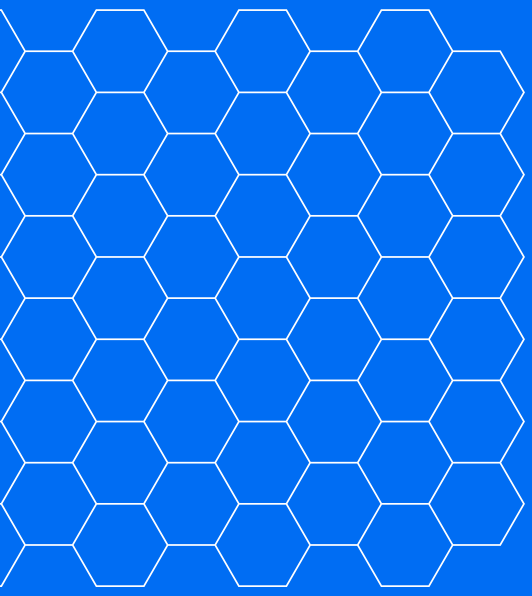
Introduction to FFF 3D printing



Ultimaker

Introduction to FFF 3D printing

Contents



EN 11/2019 v2.00

Introduction

Fused filament fabrication (FFF) 3D printing is an ideal technology for creating prototypes and end-use parts. But it's important to understand how design can effectively leverage the technology. Even small design optimizations can result in stronger prints, less material wastage, and reduced print time, labor, and costs.

This guide is for people who are familiar with CAD and CAM processes, but want to know more about 3D printing. It explores industry best practices and details how to reduce print time and errors.

FFF basics

FFF 3D printing is an additive manufacturing technology that builds an object layer by layer, by extruding thin layers of material. It can be used to create robust models and parts using engineering-grade thermoplastics. Because parts are printed layer by layer, FFF offers greater design freedom.

After designing your part, you'll need to prepare it for 3D printing. This is done by slicing software such as Ultimaker Cura, which delivers a user-friendly 3D printing experience. Ultimaker Cura streamlines the CAD and CAM workflow, by integrating with industry-standard software for maximum efficiency. It provides optimized, expert-tested print profiles, allowing new users to begin FFF printing with minimal hassle.

Reduce warping

Plastics expand slightly when printed due to heating, then contract as they cool down. If the material contracts too much while cooling, it causes the print to deform or curl. This is known as warping. To limit warping, you can adapt your design or alter its print settings in your slicing software.

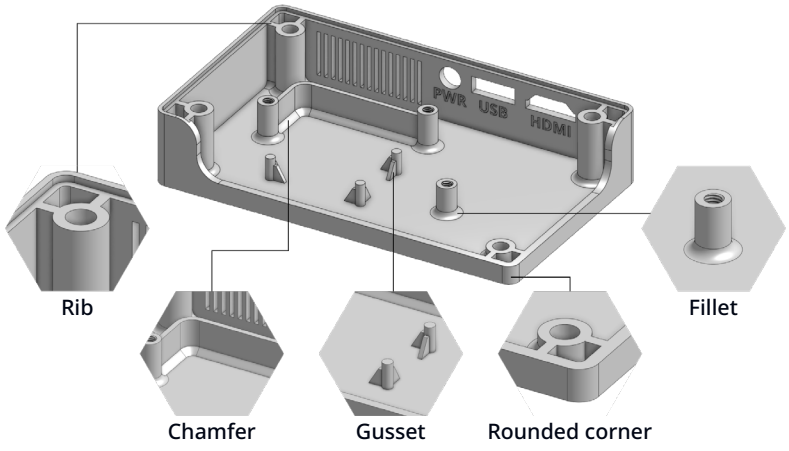
Selecting specific material profiles

Thermoplastics are proven FFF materials, with various properties that may affect build plate adhesion, which is crucial to prevent warping. However, some materials shrink more than others. For example, PC has a higher shrinkage than PLA, so is more likely to warp during the printing process.

Ultimaker Cura includes preconfigured material profiles for each Ultimaker material. Over 400 slicing settings are optimized to prevent common print faults such as warping, under-extrusion, delamination, and pillowing. Print profiles automatically select the ideal adhesion method per material. Hardware considerations, such as fan speed and build temperature, are adjusted to achieve the most reliable results.

Adapting your design for printing

Minor design adjustments can make a significant difference to print success. Large, flat areas and sharp corners are more likely to warp. But design features such as ribs, chamfers, gussets, rounded corners, and fillets reduce the chance of warping during printing.



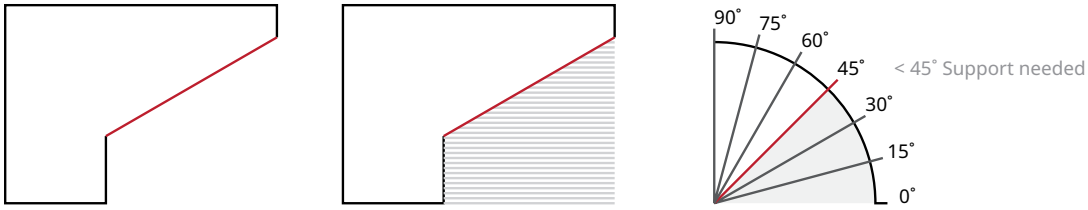
Useful design features for FFF

Changing the orientation of the part during preparation – so that the bottom has less surface area and fewer sharp edges – can also reduce warping. Part orientation can also dramatically change the part's strength and will be discussed in more detail further on.

Adding support

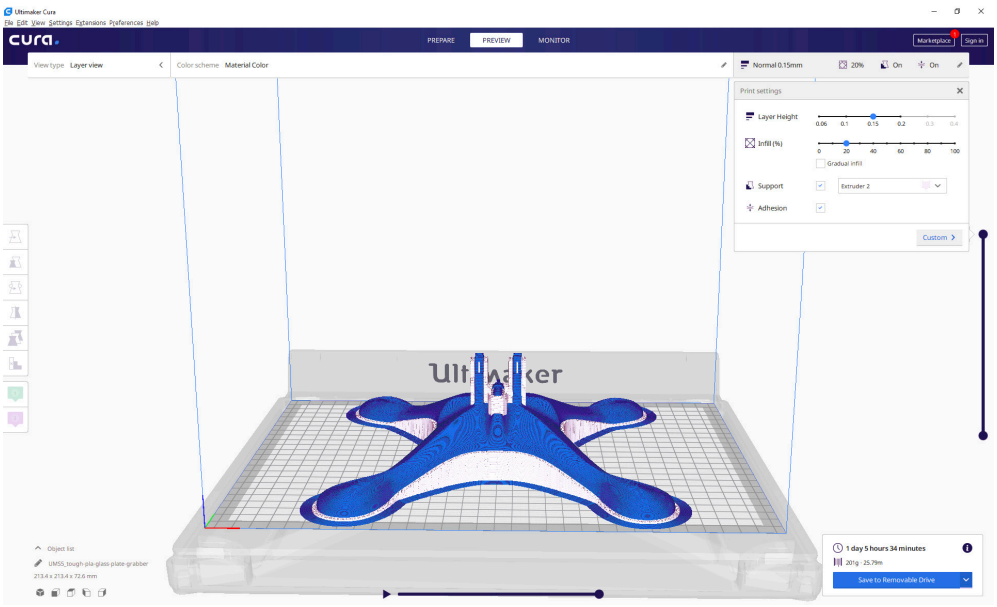
Unlike traditional manufacturing techniques, 3D printing enables large overhangs, undercuts, and interlocking parts. However, when designing for FFF, it is important to remember that material cannot be deposited onto air. Every layer must be printed on top of underlying material.

Overhangs are areas that are only partially supported by the layer below, or not supported at all. There is a limit to the overhang a printer can produce without support material. For FFF, this angle is approximately 45 degrees.



Ultimaker Cura is able to detect which areas of a model need support material and generate it automatically. When printing is complete, support material can be removed during post-processing.

Using water-soluble support material leaves the printed surface unmarked, and extends the capabilities of FFF, allowing nested and moving parts to be fabricated in a single print. Ultimaker Breakaway is another support material, best suited to designs with flat overhangs and water-sensitive build materials. It prints with reliable adhesion and is easy to remove after printing.



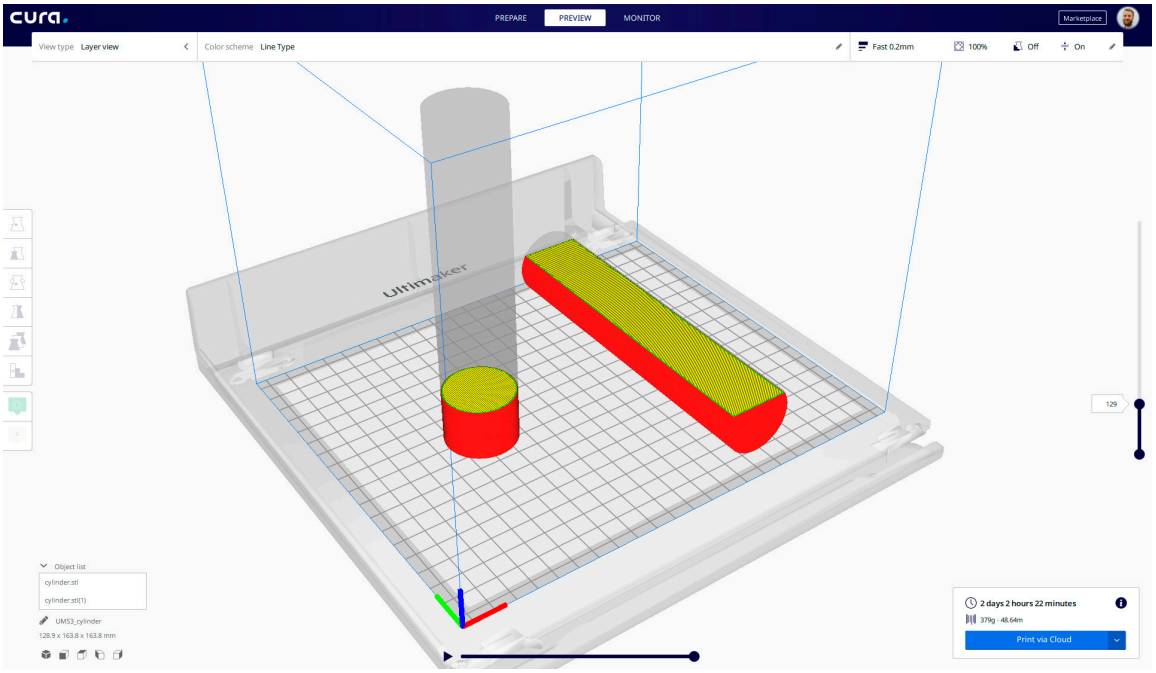
This part uses a brim for adhesion and water-soluble support material. Ultimaker Cura can automatically generate these structures where needed

Part orientation

Part orientation plays a critical role in the quality of a printed item and impacts print accuracy, manufacturing time, strength, and surface finish.

Your printer's build volume limits the maximum part size that you can print. For example, the Ultimaker S5 has a volume of 330 x 240 x 300 mm. So to print a part with a base of 300 x 300 mm, you'd need to alter the orientation of the part. You may also want to adjust a part's orientation to reduce support material. Ultimaker Cura is compatible with a plugin that calculates the optimal orientation for your print, for the most efficient use of support material.

To illustrate – let's say we print two tubes. The first has a vertical center axis. A 3D printer would construct this tube as a series of circles layered on top of one another, resulting in a cylinder with a smooth outer surface. If the same tube is oriented horizontally, it will print as a series of rectangles of different widths, layered on top of each other. This would produce a tube with a 'stepped' surface. The surface touching the build plate would have a flat edge, unless printed using water-soluble support material such as Ultimaker PVA.



Part orientation affects the reliability and quality of the print

Part orientation impacts:

- Surface quality
- Strength (FFF printed parts are stronger in the XY direction than in the Z direction)
- Print time (more layers can affect print duration)
- Required support material (which affects print time and cost)

Printing smaller parts

It is possible to print very small parts and structures with FFF, but there is a limit to how small or narrow you can go. For example, printing thin walls can cause:

- Warping, as material is more likely to bend and buckle before it can cool
- Delamination, as it is difficult for the printer to deposit a new layer onto the existing body
- Difficult post-processing, due to the part's fragility

Minimum wall thickness

The recommended minimum thickness of unsupported walls is 1.2 mm. Walls thinner than this may become distorted or detach from the model during printing. For supported walls (connected to at least two other walls), the recommended minimum thickness is 0.8 mm.

In general, wall thickness should be a multiple of nozzle size. Wall strength also depends on the type of filament, layer thickness, and the wall's size and geometry. Integrated support features (e.g. ribs) can help to increase stability.

Fillets for stronger prints

Some structures need to be printed with supports – but this can dramatically slow the print process. Fillets reduce print time, while increasing surface area at points where one structural feature merges with another. This reduces stress during printing and adds strength to the part.

Printing holes and cylinders

3D printing can result in undersized printed hole diameters. When accuracy is important (e.g. when connecting separate parts), it is best to design larger holes, or to enlarge holes by machining during post-processing.

Ultimaker Tough PLA is a machinable material that has the toughness of ABS and reliability of PLA, so is well-suited to most post-processing methods.

- For holes up to 10 mm in diameter, best practice is to add 2-4% tolerance, with a smaller percentage for larger diameters
- The recommended minimum hole diameter is 2.0 mm. The deeper the hole, the higher the minimum value should be
- Remember that a hole can always be enlarged, but can’t be made smaller
- Holes with a vertical center axis will print more accurately
- Holes with a smaller diameter may close over or deform during printing

Use ACME threads

When designing screw threads, avoid sharp edges and 90-degree angles, as they can create stress areas in plastic parts. ACME threads work well with FFF. Their screw thread profile has trapezoidal outlines, is easy to manufacture, and offers high strength.

- The recommended minimum size for the thread is 0.8 mm. Small threads produced with FFF won’t work with holes smaller than 3 mm

Modular design

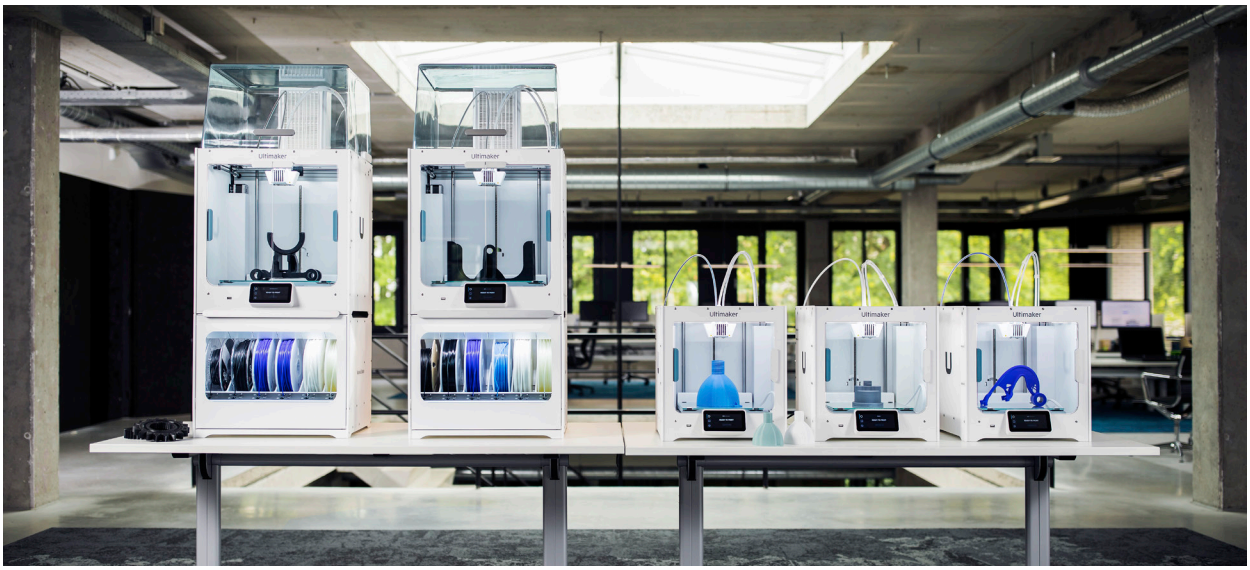
To reduce support structures, you can design a part in modular pieces and assemble them after printing. Modular design is also beneficial when you want to:

- Create parts larger than the printer’s build volume
- Print fragile parts in another orientation, to produce a stronger part

Multiple 3D printers can simultaneously create the parts of a single assembly. This is faster than printing parts sequentially on a single printer. Ultimaker Connect manages multiple 3D printers from a single interface, making it easy to add printers as needed.

Find a reseller

Ultimaker's global network of dedicated service partners is ready to help you find a 3D printing solution to give your business the edge. Browse our worldwide network of resellers at ultimaker.com/resellers.



Explore more 3D printing knowledge

Learn more from industry leaders and experts, and compare the specs of our 3D printers, on the Ultimaker website

Read more 3D printing resources

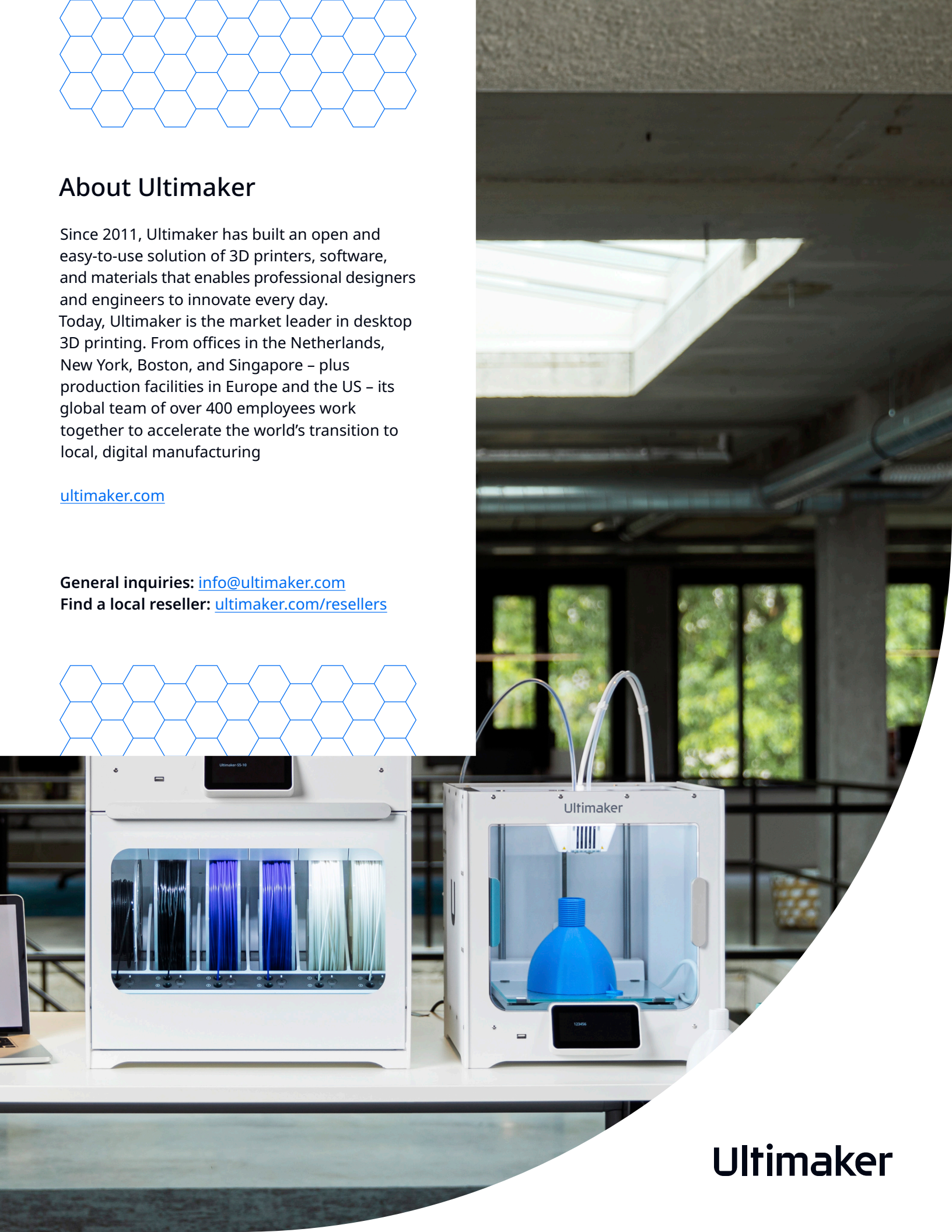
Request a quote from Ultimaker

About Ultimaker

Since 2011, Ultimaker has built an open and easy-to-use solution of 3D printers, software, and materials that enables professional designers and engineers to innovate every day.
Today, Ultimaker is the market leader in desktop 3D printing. From offices in the Netherlands, New York, Boston, and Singapore – plus production facilities in Europe and the US – its global team of over 400 employees work together to accelerate the world's transition to local, digital manufacturing

ultimaker.com

General inquiries: info@ultimaker.com
Find a local reseller: ultimaker.com/resellers



Ultimaker