From the CAD design for the 3D printed component

A practical guide to 3D printing

Markforged

Mark3D
THE Markforged Experts

Mark3D
THE Markforged Experts

Function meets innovation

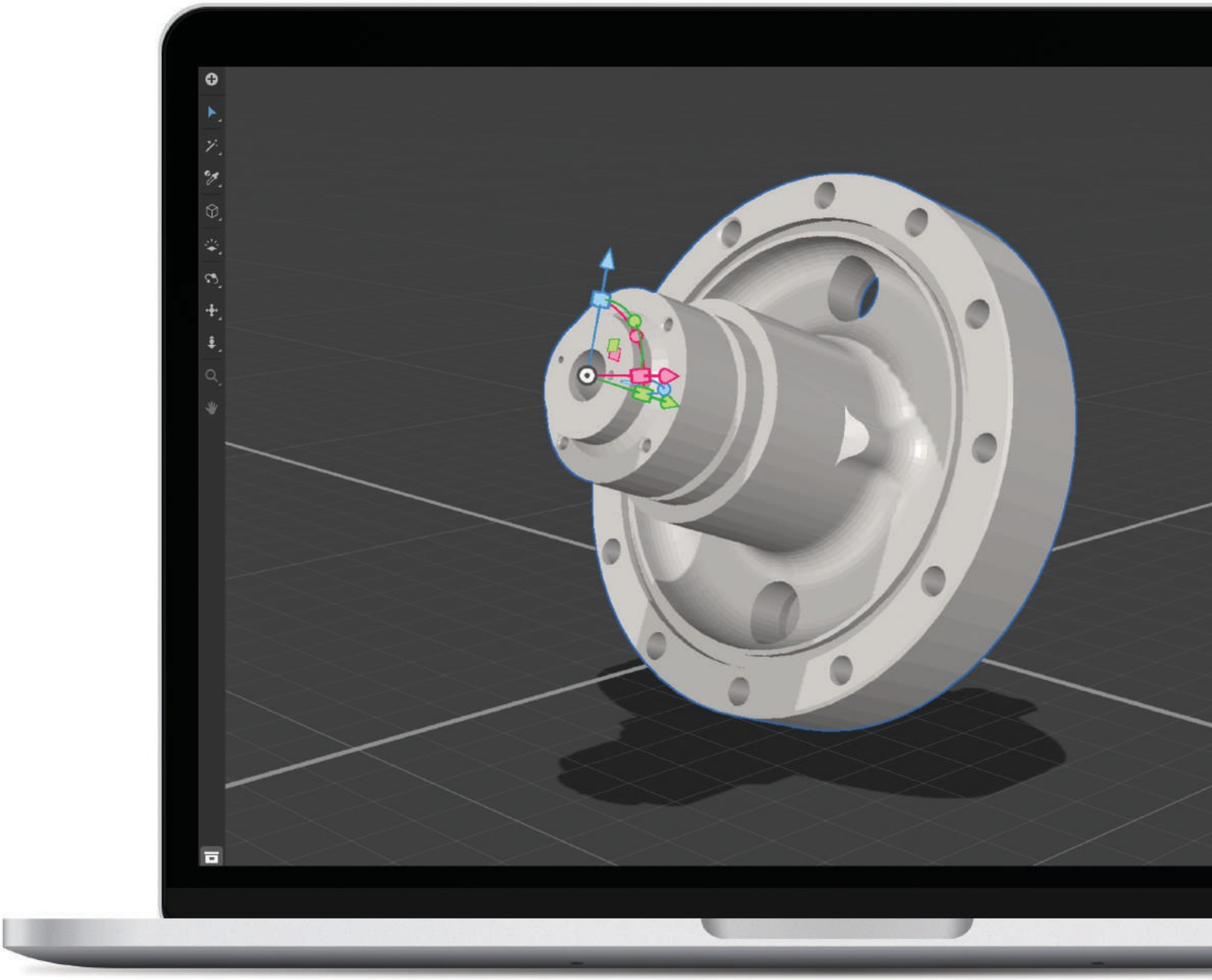
3D printing software is the most important interface between user and machine. Modern software for additive manufacturing can be used innovatively in
production processes - but their main function is to take you, the user, from design to finished part.

This white paper looks at the individual steps from design to printed component. We show the key factors and the functionality of 3D printing software for each section. Although this guide is written in the context of Markforged's FFF and CFR technologies, the considerations also apply to other printing technologies.

Design

The first step on the way from design to part is to create the part design in a CAD platform. We recommend our design guide for 3D printing with composite materials, which provides basic guidelines for printing and
contains the component design. You can then export the CAD design as an STL file.

When designing for 3D printing, it is best to use established CAD software with proven STL export functions. These can be found in almost all commercially available CAD programs as well as in the more popular open source or hobby packages. If you are not sure which software you should use, check the
STL export options.



Export

What is an STL?

STL is the worldwide industrial standard file type for 3D printing. STLs are a full-body representation of a part. The 3D printing software can analyze these and convert them into print commands.

STL files describe a series of triangles that approximately represent the surfaces of a 3D model. All the triangles together (usually millions of them) form a mesh that describes the three-dimensional geometry. You can import this mesh into slicing software, such as Markforged's Eiger™ software platform, in preparation for 3D printing.

STL parameters

The settings you select in your CAD software when creating and exporting an STL file can have a significant impact on the quality, dimensional accuracy and surface finish of your 3D print. There are two main parameters that influence STL generation:

+ **Chordal tolerance/deviation -** controls the dimensional accuracy of the STL compared to the design.

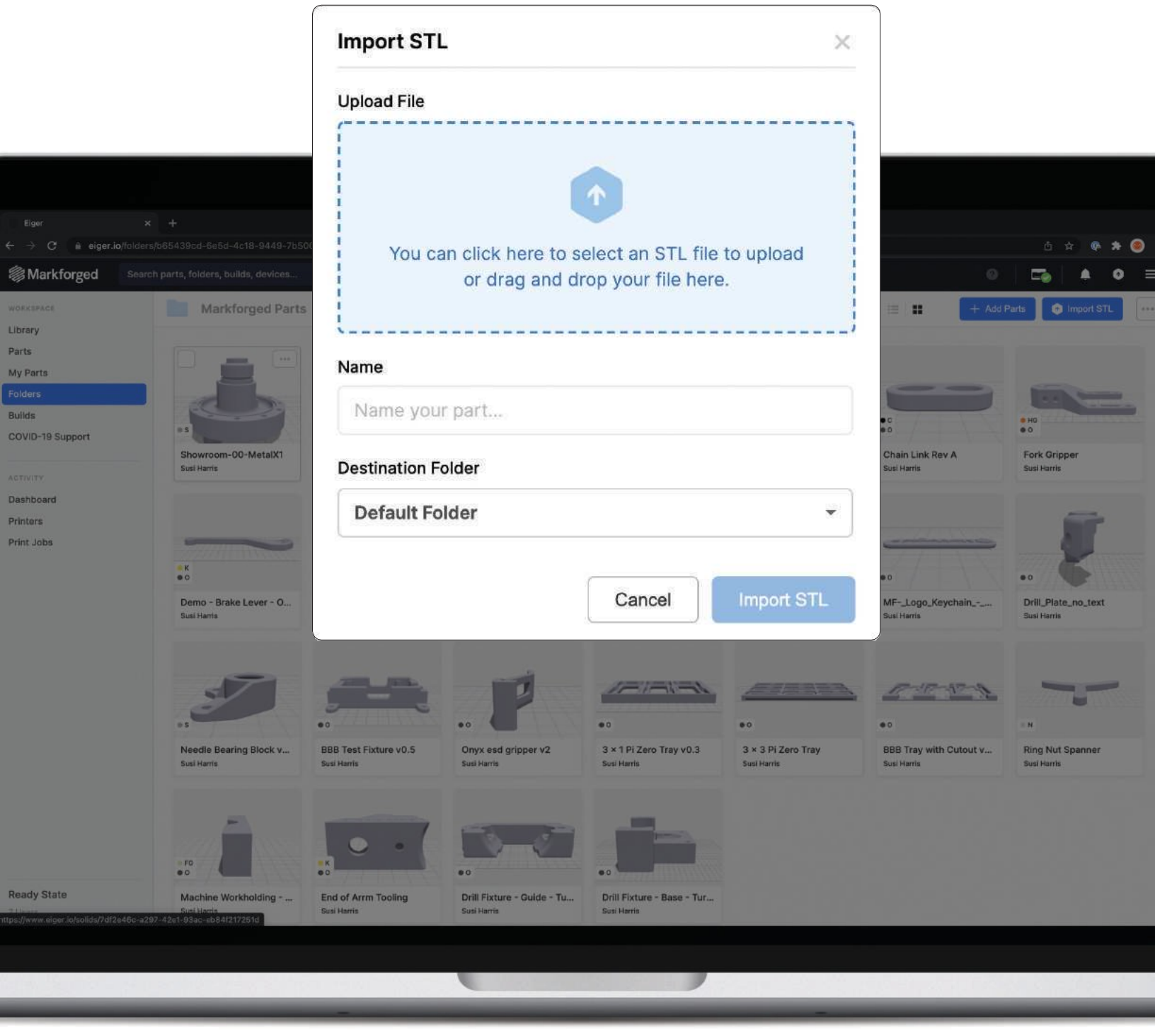
+ **Angle tolerance/deviation -** controls the maximum angle that is permitted between the vectors of two neighboring triangles in the mesh. This "refines" the mesh with a higher resolution than the chordal tolerance would otherwise allow.

Although a higher resolution produces smoother and more accurate models, at a certain point the resolution of the STL file may exceed the machine accuracy of your 3D printer.

You will find additional settings in some CAD programs. We recommend that you keep the default values unless you have a special reason to adjust them. In most cases, you will achieve the best results if you select the "fine" setting and binary encoding when exporting in order to minimize faceting.

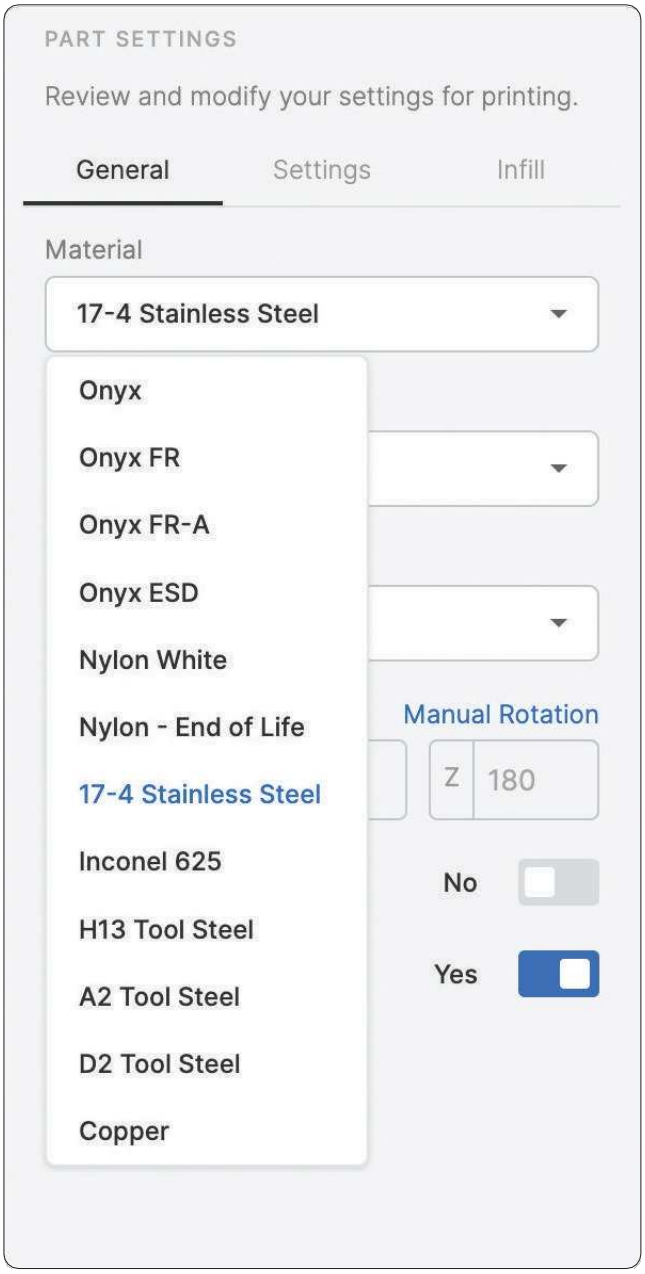
Import

Once the STL file has been exported from the CAD program with the appropriate parameters, it is imported into a cloud-based or local slicing software program.



Configuration

There are various configuration options when importing the STL file into the slicing software. With 3D printing, the manufacturing process can also be controlled: Unlike conventional manufacturing, individual design options are available even after the design has been completed.



The standard settings are usually suitable and make it possible to quickly prepare files for printing. More detailed settings and layer-by-layer preparation can control the properties of composite or metal parts. For example, in the case of composite parts produced with the FFF and CFR
material from Markforged, or for metal parts that are printed with our metal FFF. Please note that part settings refer to the finished part and the print settings determine the printing process.

Metal parts

Metal FFF is often less configurable than CFR composite printing, but important aspects can still be configured.

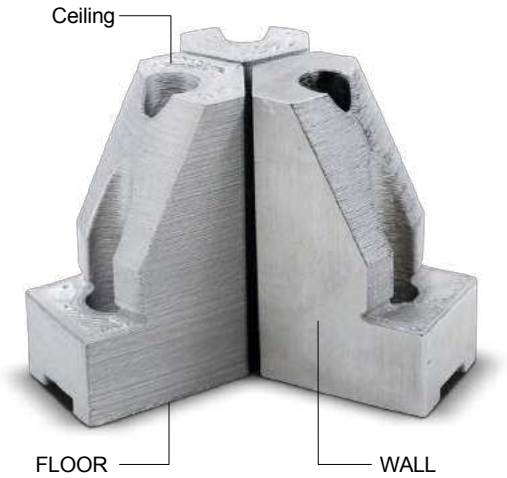
Component settings

The component settings determine the quality of the finished part, while the print settings control the printing process.

+ **Material -** When slicing a metal part, the first question is what material to use. Our Metal X™ offers six different materials: 17-4 PH SS (stainless steel), H13 tool steel, A2 tool steel, D2 tool steel, Inconel
625 (nickel-based superalloy) and copper.

+ **Infill -** Metal FFF parts have a closed-cell grid infill as standard. You can customize the infill or print solid parts. With the grid infill
you can control the geometry and density of the grid.

+ **Walls, floors and ceiling layers -** You can dynamically control the wall thickness of your parts by changing the number of walls, floors and ceilings. With a
The thickness of the walls, floors and ceilings of a solid component is not relevant.



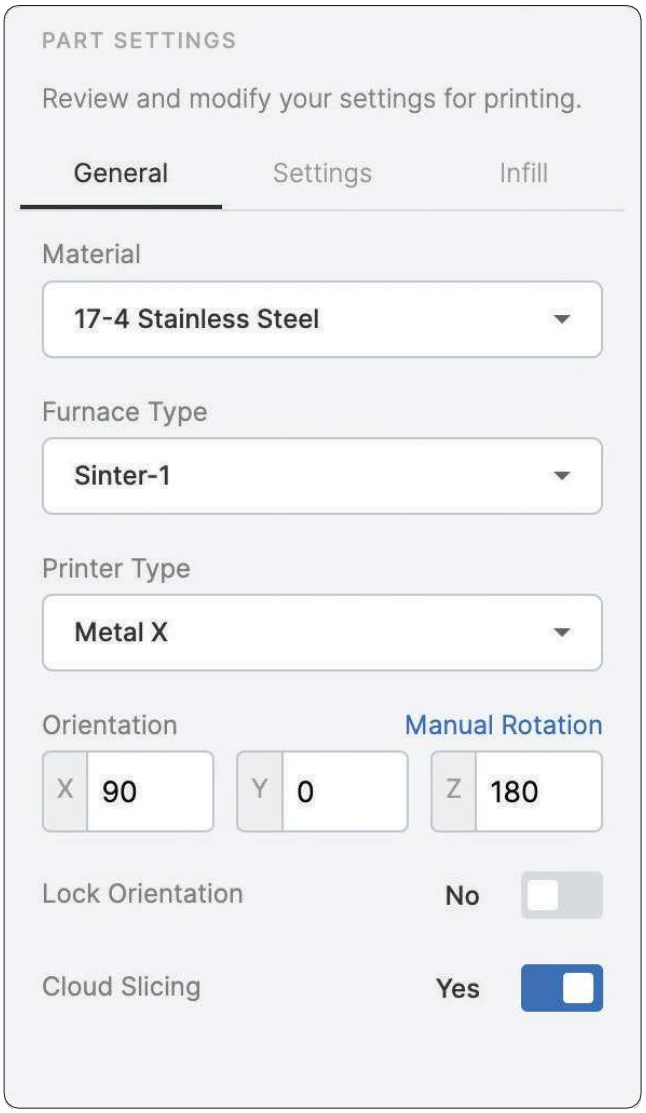
Print settings

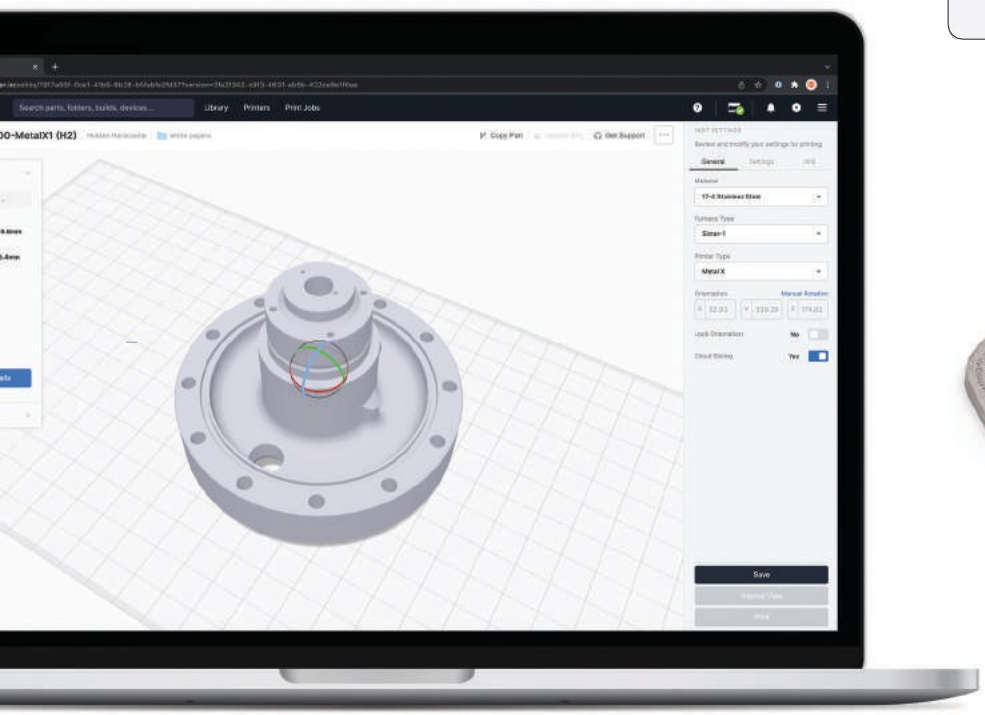
+ **Alignment - The** most important print setting is the alignment of the component. This is determined by clicking on the surface that should be parallel to the print plane. Markforged recommends selecting a large, flat surface and the avoidance of protruding elements.

+ **Supports and rafts -** For FFF metal parts, the slicing software automatically creates supports for protruding surfaces. Eiger adds a base and supports to prevent warping during printing and to ensure accurate shrinkage. Eiger adds a layer of ceramic material so that the parts can be easily separated from the supports after sintering.

  You can activate supports and rafts as required. Supports can greatly improve the quality of the component, even if they increase the printing time slightly. You can print supports as a whole body or in individual sections. Splitting supports is ideal for some complex printed structures so that the supports can be easily removed.

+ **Layer thickness -** Low layer thicknesses provide a higher resolution but slow down printing. Higher layer thicknesses are printed faster, but result in parts with lower resolution. If there are no deviating reasons, we usually recommend the standard settings.



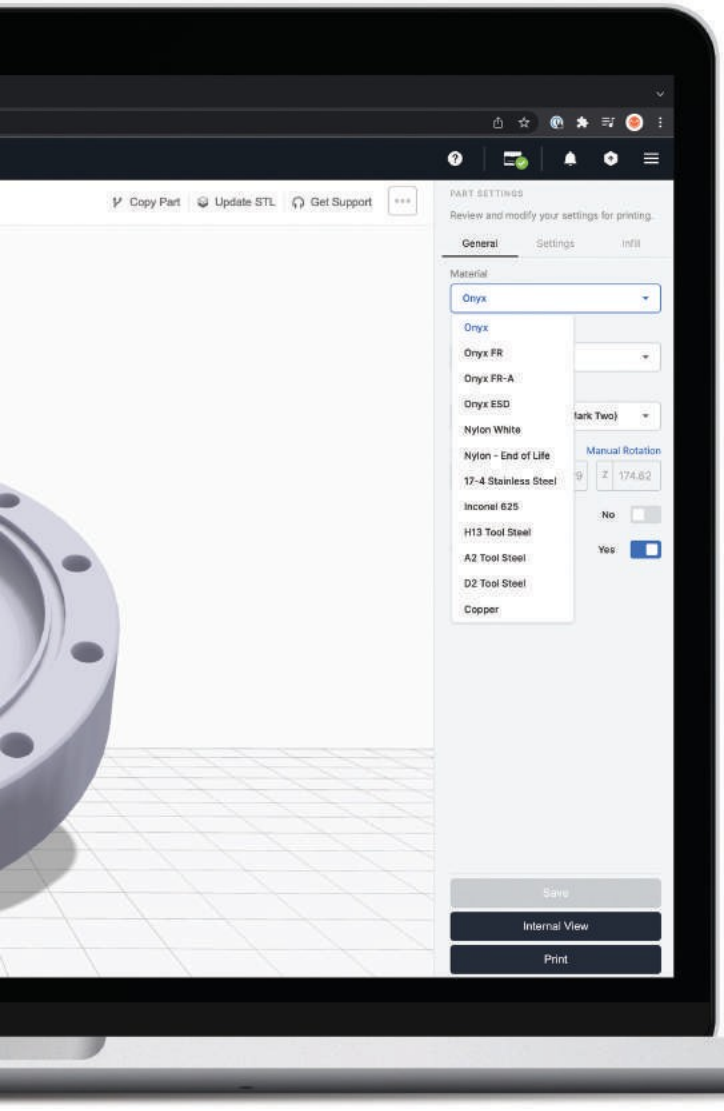


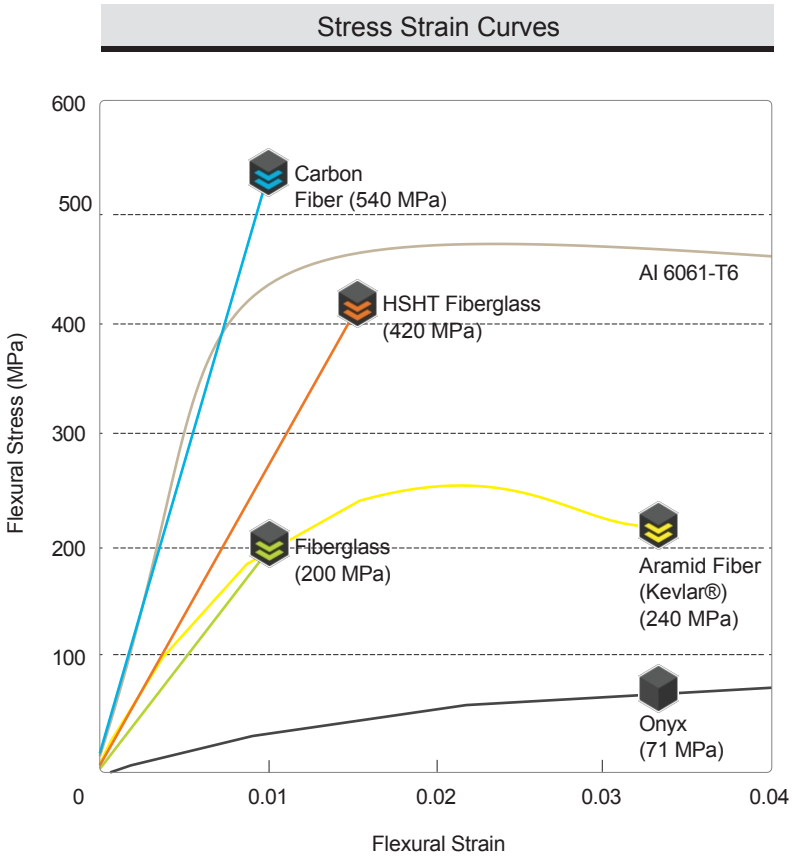
Composite parts

With the use of fibers in CFR, composites offer more configuration options. In contrast to FFF printing, CFR creates more individual options.

Component settings

Materials - As with metal, the choice is crucial, but here it is more complex. Our composite printers can print two materials: conventional FFF filament
and continuous fibers.





Recommended steps:

1. Choose a composite base filament. Markforged offers three main types of short fiber-filled nylon: Onyx™, Onyx FR™ (flame retardant) and Onyx ESD™ (ESD compliant).

2. Select a continuous filament. You can choose not to reinforce a part or use fibers. Markforged offers carbon fibers,
Glass fibers, aramid fibers (Kevlar®)* and high-strength, high-temperature-resistant glass fibers (HSHT).
Reinforced parts have the strengths of the respective fiber material in terms of bending and deformation forces. Consider which strengths your component requires: Resistance to breakage or permanent deformation (bending stress) or to elastic deformation (bending strain).

**Dupont™ and Kevlar® are trademarks and registered trademarks of E. I. du Pont de Nemours and Company.*

Reinforcement strategy

You can design the fiber reinforcement individually. This can significantly change the component properties during use.

In the standard setting, several layers at the top and bottom of the part are reinforced with fibers.

You can also change many details when reinforcing parts. The two most important factors are

+ **Which layers are amplified.** Individual layers or groups of layers.

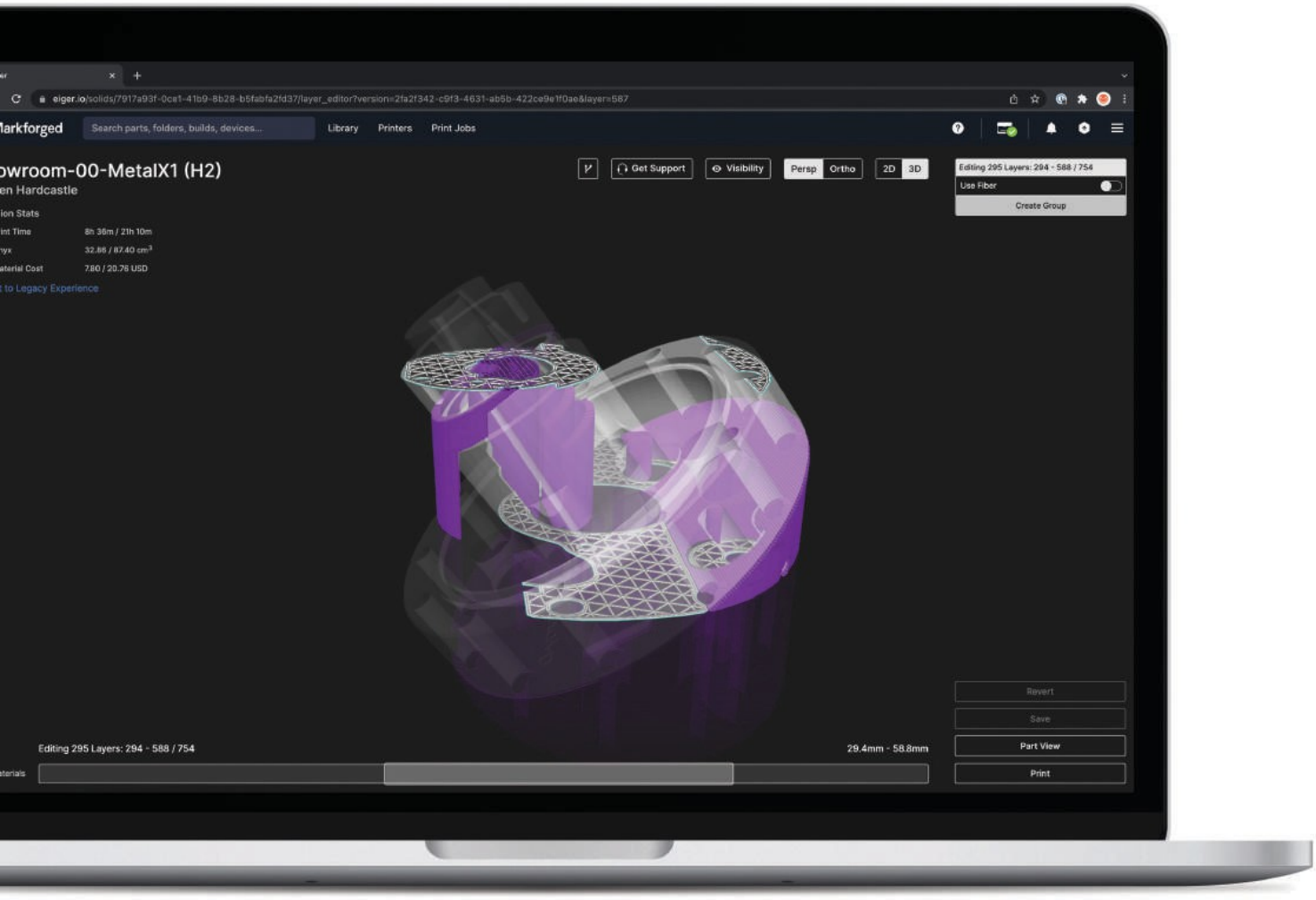
+ **How the layers are reinforced.** Choose one of several strategies for each layer and select the orientation and number of fibers laid.

Infill, walls, ceilings and floors

The settings for infills, walls, ceilings and floors are customizable. A geometric infill can save costs and time and be more accurate. A higher infill density can improve buckling resistance, a reduced density can improve the strength to weight ratio.

With walls, ceilings or floors, your component can become more resistant to abrasion, cuts and impacts. A solid infill improves the compressive properties and offers the option of adding holes and threads if required.

.



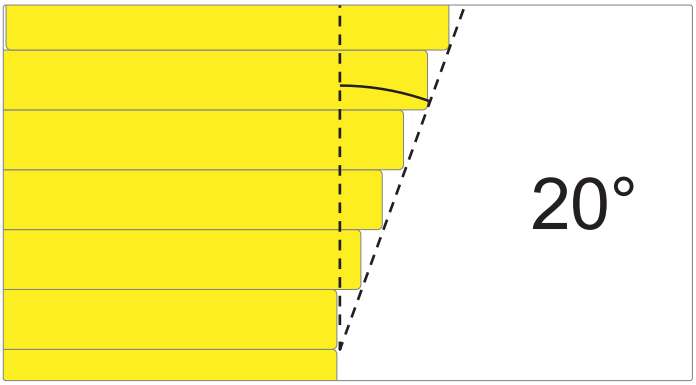
Print settings

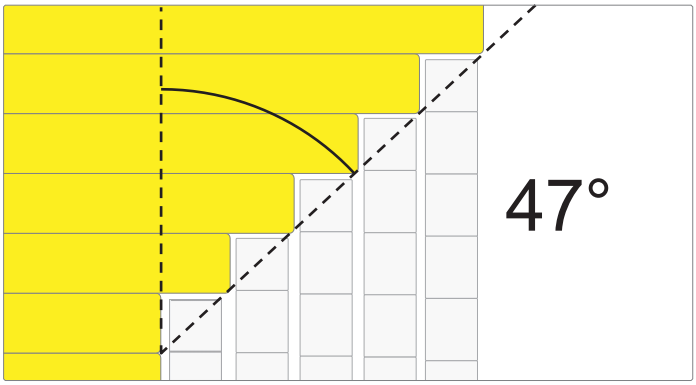
+ Alignment - The settings for CFR composite printing are handled in a similar way to metal printing.
The alignment also determines the areas where CFR is added. It is therefore important to consider where the reinforcement is located when positioning.

Normally, the alignment should minimize overhangs, while the surface area in contact with the print bed should be as large as possible. This reduces the number of supports and improves adhesion to the print bed.

+ Supports - Slicers create supports from a certain angle. For example, Eiger automatically creates supports for any surface that is more than 45°. The supports can be activated depending on the material and shape. High-quality nylons with sliced fibers,
such as Onyx, allow overhangs of up to 60° due to their resistance to thermal shrinkage.

Threshold angle

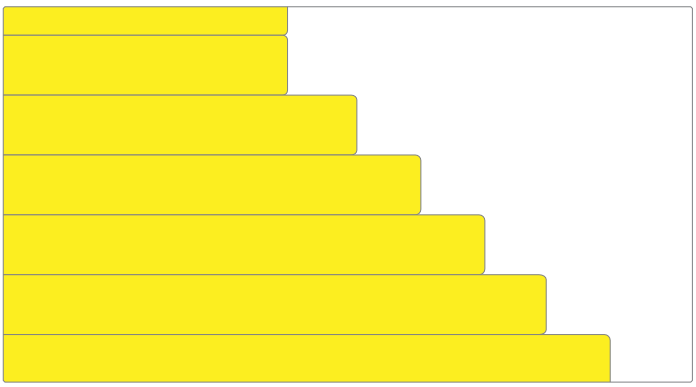


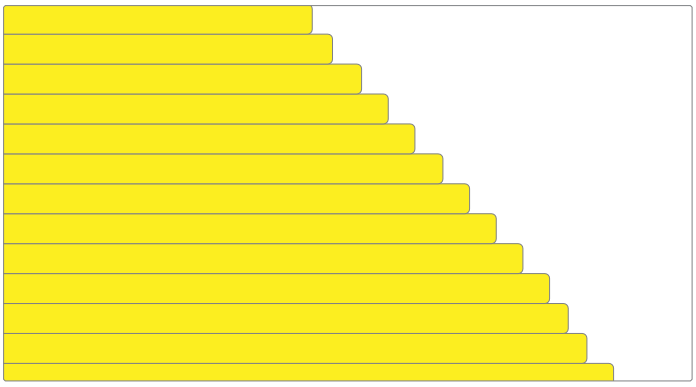


+ Layer thickness - Slicers allow you to reduce the layer thickness of composite parts.
and thus change the resolution. This primarily affects the surface quality. A higher resolution requires a lower printing speed.

For faster printing in design quality, the layer heights in individual areas (such as infill) can be adjusted on some slicers. Examples of this are Eiger's Turbo Infill and X7 Turbo mode (4-fold increase for the latter). This setting is ideal for designs or components where sophisticated features are less important.

Layer thickness



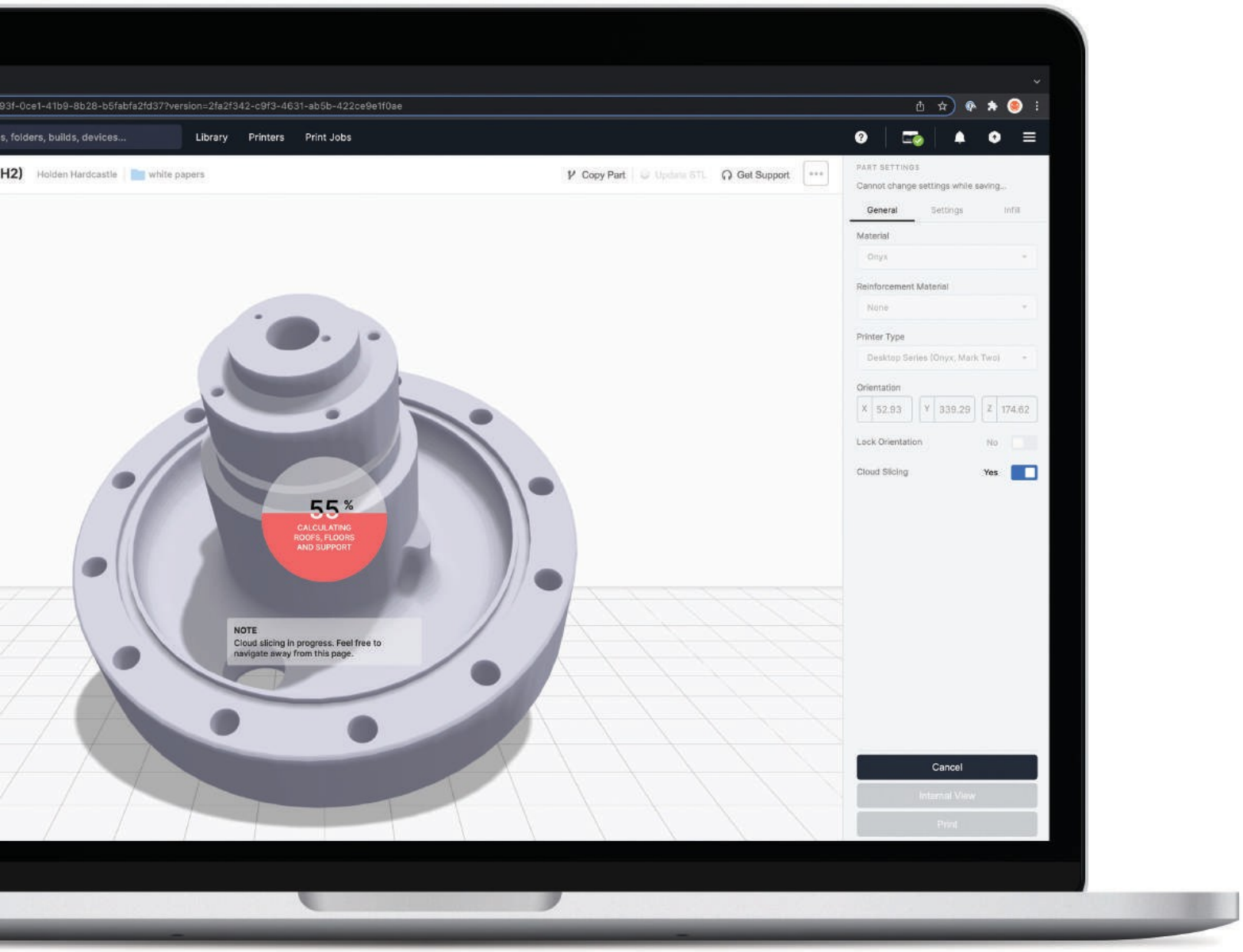


Slicing

The slicing engine now converts the part and the print settings into commands for the printer. It is best to use software with printing algorithms that have been refined and optimized over the years so that you get repeatably accurate and stable parts.

After slicing, the 3D printing software, such as Eiger, provides you with detailed information about your component:

+ **Component costs -** The costs for plastic and fibers. For metal, the costs for metal and ceramic surfaces are displayed.
+ **Dimensions and weight -** The dimensions and weight of the finished part.
+ **Print time -** For metal parts, the debinding and sintering time is displayed so that you can get an idea of the time per part. For composite parts, the printing time is displayed.



Layout

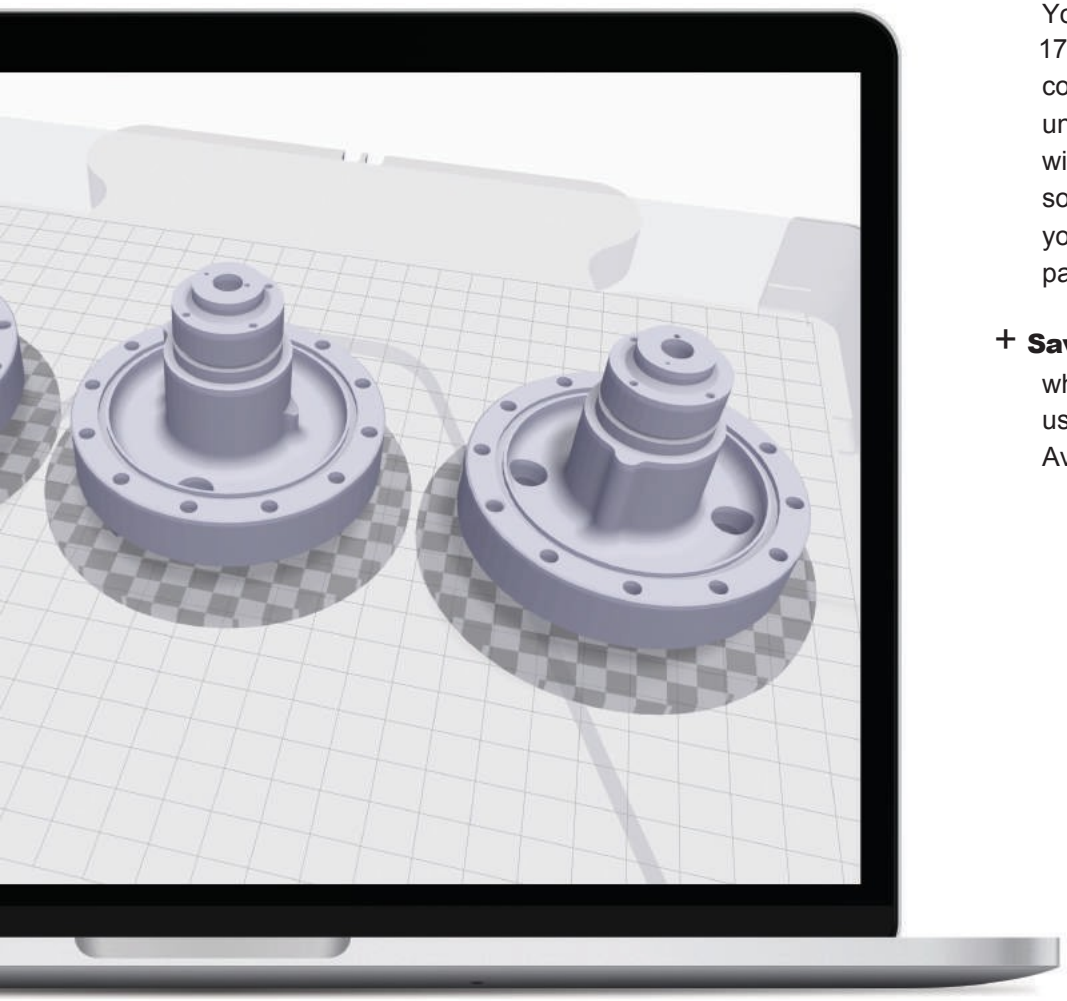
Several parts made from the same materials and with the same layer heights can be combined in the print. At the same printing speed, several parts are created immediately in one build job. Such "builds" are useful tools, but not without disadvantages.

Reasons to use "Builds":

+ It's all about convenience and control of print planning.

+ By combining several parts in one build job, you can reduce the maintenance required for your printer. With four parts in one build job, you save three print cycles.

+ To make the printing process fit better into your schedule, you can, for example, combine two 24-hour prints with a 16-hour print.
and make the most of a weekend. Such construction jobs optimize uptime and scheduled maintenance for your printer.



Reasons not to use "Builds":

+ "Builds" can increase the risk of errors. If the build job fails, every part is unusable. In some cases, complex parts should be printed separately.

Recommendations

+ **Restrictions -** As mentioned, the parts in a build job must have the same print material and the same layer thickness. With FFF printing of metals, this is quite simple: Print
You only use 17-4 parts with other 17-4 parts. With composites it is more complicated: you can only print an unreinforced part with a reinforced part with the same layer thickness. Your software, such as Eiger, should inform you if a build job contains compatible parts.

+ **Save " Builds" -** This is ideal for users who regularly create "builds". You can use this to simplify the entire process. Avoid reprinting.

The pressure

The last step:
press the push button.

Become a partner of Markforged, a pioneer in 3D printing software

Eiger from Markforged is the leading software for digital manufacturing and accelerates the further development of your company. With the first networked, secure platform for additive manufacturing, companies can use any
Prepare, print, manage, scale and produce parts as required with ease.

With its unique approach, Eiger provides an integrated cloud-based platform that allows you to print both metal and carbon fiber parts directly from your browser. Eiger provides easy access to slicing and connects all Markforged 3D printers. Users can log into their account to manage the print schedule, assign jobs and track print progress in one simple platform.

Markforged's 3D printing slicer software allows users to track the real-time status of any printer from anywhere in the world. The next generation of manufacturing requires more innovation in production. IoT, big data and cloud technologies are the key elements to making manufacturing smarter. Eiger harnesses all these new technologies and integrates them into one.

Learn more about Eiger on
https://www.mark3d.com/de/eiger-software/

We look forward to seeing you!

Mark3D GmbH
Rodenbacher Street 15
35708 Haiger

Phone **07361 63396-00**
E-mail **markforged@mark3d.de**
Internet **www.mark3d.de**

Mark3D Switzerland GmbH
Book 43
9322 Egnach TG

Mark3D UK Limited
The Innovation Centre, 1 Devon Way
Longbridge, Birmingham, B31 2TS

Mark3D Benelux BV
Brabantlaan 3D
5216 TV, 's-Hertogenbosch

© 2022 MarkForged, Inc. & Mark3D GmbH

All prices quoted are non-binding and are subject to VAT.
We accept no liability for any of the above information. Only the original documentation of the manufacturer Markforged Inc. is relevant for binding performance commitments.