TIGITAL® 3D-Set

DESIGN GUIDELINES

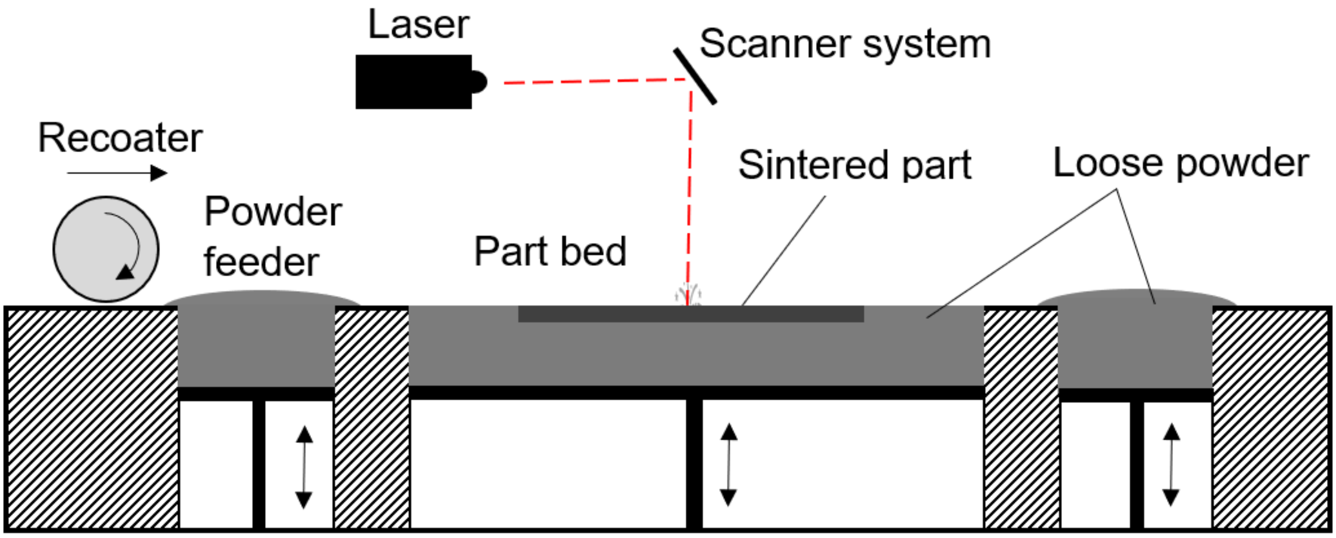
CONTENT

INTRODUCTION

How does SLS 3D-Printing work?

Selective Laser Sintering (SLS) is an additive manufacturing process similar to Fused Deposition Modelling (FDM). Unlike FDM, which utilizes materials in the form of filaments, SLS utilizes materials in the form of **powders**.

The powder is sintered and **fused using a laser beam**, then recoated to deposit a new thin layer of powder, which is then sintered again. This process continues until the desired shape is created.



As soon as the printing is finished, the material needs to cool down. As a rule of thumb, conventional SLS materials (like polyamide) require a cooling time similar to the printing time. One of the benefits of thermoset polymers is a reduced cooling time – typically **half of the printing time is sufficient**.

The powder cake is then be removed from the printer and the finished parts depowdered from the remaining loose powder. The powder removal process works best with a combination of **compressed air, sandblasting and brushes**.

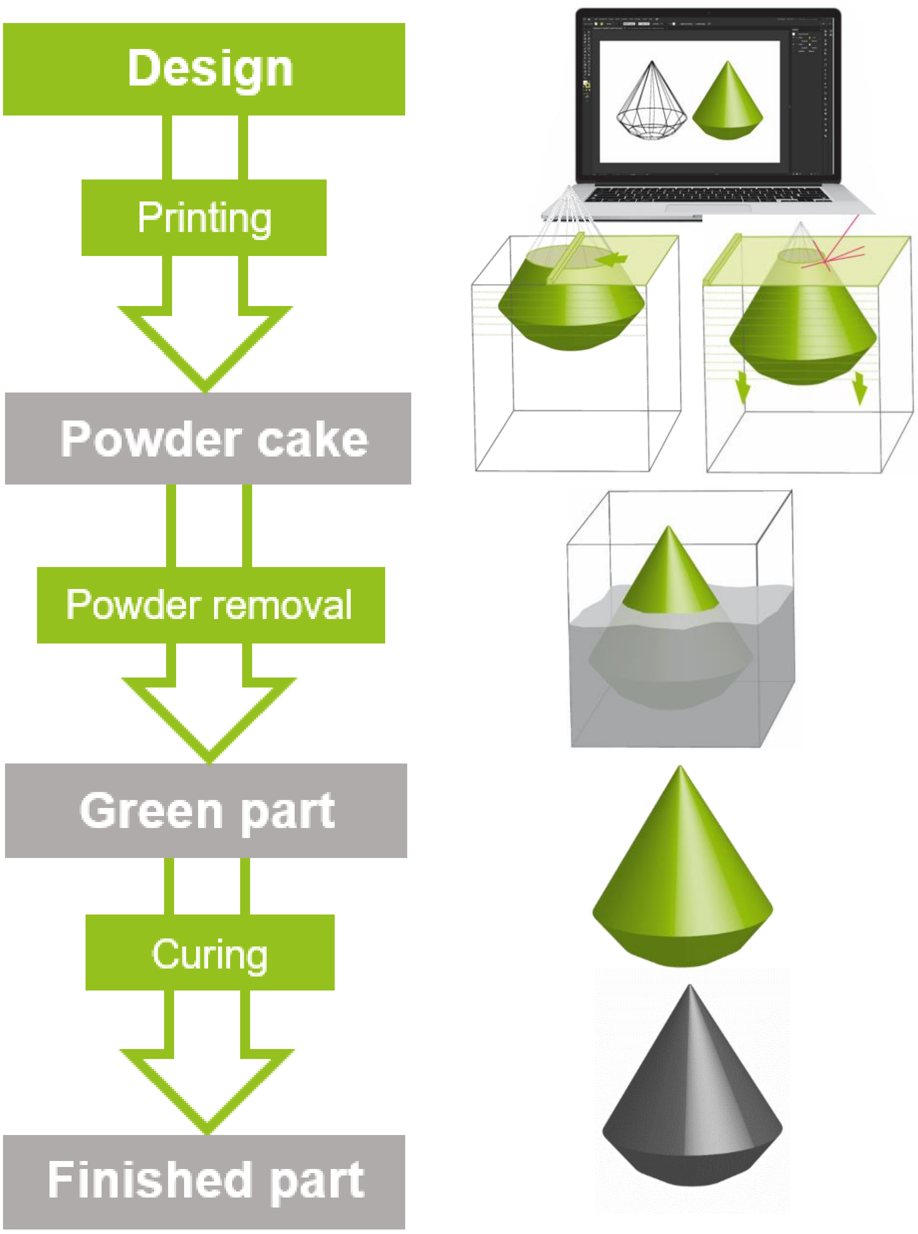
A major benefit of SLS is that even the most complex structures can be created, at a very high resolution and accuracy, without the need for support structures.

INTRODUCTION

SLS 3D-Printing with thermoset polymers

The main difference between thermoset polymers and conventional materials, apart from the mechanical properties, is the requirement for hardening the sintered parts (which are referred to as green parts).

This step is necessary for **crosslinking** the resin which ultimately develops the full potential mechanical properties of the material. For this reason, **green parts are more fragile** than the finished product and should be handled carefully.



GENERAL ADVICE

While designing for SLS 3D-Printing is very similar to other manufacturing methods, there are techniques that will help to be successful with your design:

When possible, round the edges

Round edges reduce stress peaks and are generally sturdier. If you are designing protruding shapes like pins, rounding the base will improve the adherence and sturdiness.

Try to design your parts with a uniform wall thickness

A uniform wall thickness can reduce stress, improve temperature distribution and minimize warping.

Consider the part orientation

Certain shapes and structures yield different results depending on the orientation they are printed at. The reason for this phenomenon is the difference between the resolutions of the laser in the XY and XZ axis. An in-depth outlook on orientation and its effect on various structures will be provided in the following chapters.

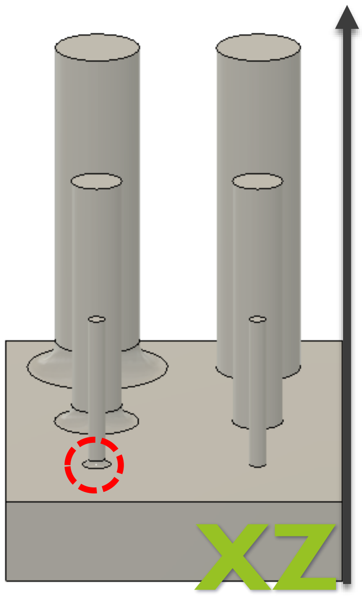
Consider the powder removal process

Even though very delicate structures can be printed using SLS, they can prove to be difficult to remove without damaging them. Keep this in mind when designing and placing your design into the printer – leaving more space between delicate structures will make the powder removal easier.

Consider the curing process

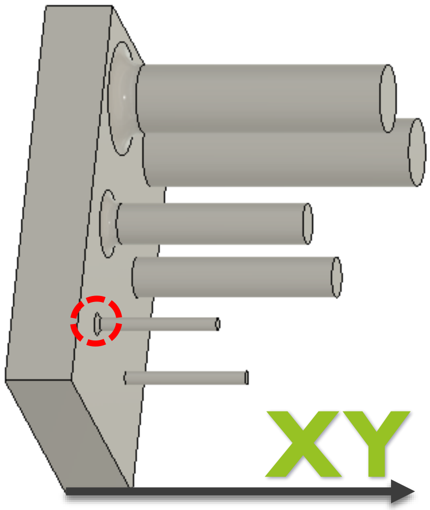
Any parts that are in contact during the curing process will fuse together. Also keep in mind that the material softens temporarily when heated up during the curing process – structures that cannot be externally supported or filled out will collapse.

PINS



The **smallest printable pin** diameter facing the **XZ or XY axis is 0.5 mm**. However, such delicate structures will be challenging to depowder. For a **sturdy structure**, a diameter **> 1.5 mm** is recommended.

Pins printed in the **XY axis** are generally **sturdier**. If handled carefully, pins that are as small as 1 mm in diameter can be depowdered successfully.



For small pin diameters (< 2 mm) **a L/D ratio of 15 should not be exceeded** – this is especially true for pins printed facing the XZ direction.

The stability of pins with a diameter **> 2 mm seems to be less affected by the L/D ratio**, a L/D ratio up to 20 has shown no negative impact on stability for pin diameters > 2 mm.

Always round the base of your pins, this will improve the adherence and stability.

To ensure good stability even for longer pins, a diameter of at least 2 mm is recommended.

If possible, print the pins facing the XY direction. This will improve the stability during the powder removal.

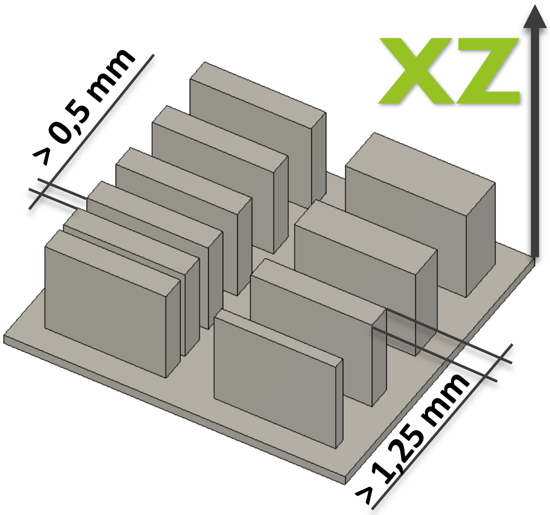
RIBS

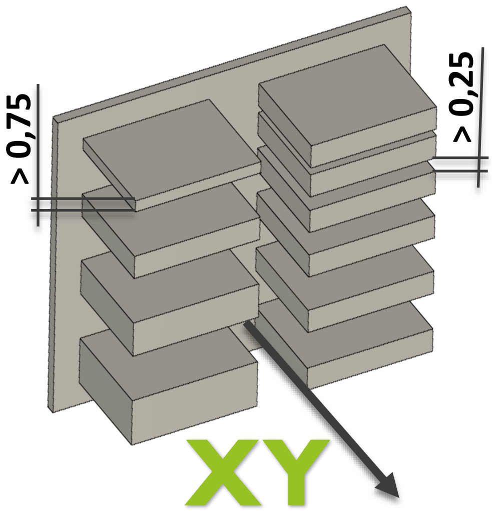
The **minimum required distance** between ribs printed facing the **XZ direction is 0.5 mm.**

The **minimum required distance** between ribs printed facing the **XY direction is 0.25 mm.**

The **smallest recommended thickness** for ribs printed facing the **XZ direction is 1.25 mm**. A thickness below 1.25 mm will result in a fragile structure that will be difficult to depowder.

Ribs printed facing the **XY direction** are **sturdier**, therefore a thickness of **0.75 mm** is sufficient for stability and powder removal.





Delicate ribs can be especially difficult to depowder. Due to the large surface area they tend to snap off easily if you blow compressed air directly at them.

The stability of ribs is significantly influenced by the part orientation – if you want to print delicate ribs, placing them in the XY direction is recommended.

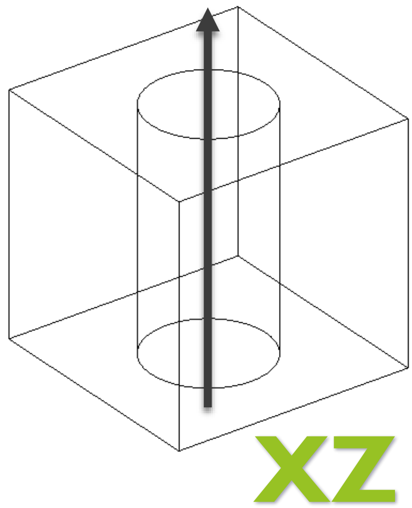
HOLES

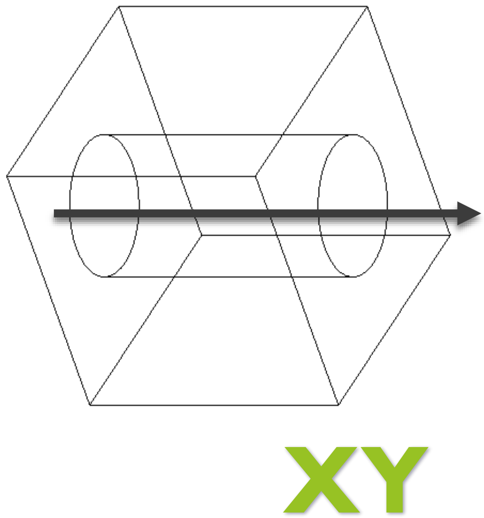
The **smallest** printable diameter of a hole placed in the **XZ direction is 1.25 mm.**

The **smallest** printable diameter of a hole placed in the **XY direction is 1 mm.**

Generally**, holes printed facing the XY direction will not have a perfectly round shape**. The reason for this lies in the nature of SLS technology – holes printed in the XY direction have their circumference printed one layer at a time, which leads to different cooling rates, ultimately resulting in a more oval shape.

Keep in mind that the part has to be depowdered, cleared of excess powder and cured. Due to this reason, there is one more limiting factor in the hole diameter, which is **the thickness of your part**.





If a perfectly round hole is required by default, place the hole in the XZ direction.

If you decide to print the holes in the XY direction, post processing may be necessary to improve the hole accuracy.

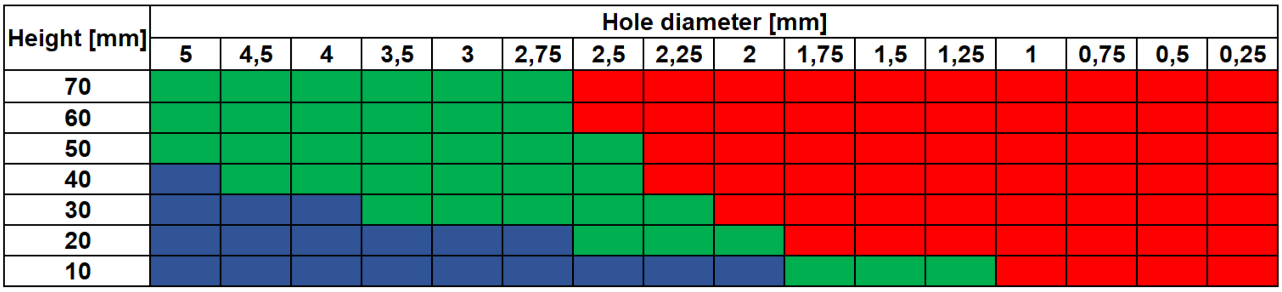
HOLES

Holes that are **smaller than 3 mm** in diameter get more **difficult to clear** out manually with an increasing part height. If you want to ensure that the hole will be clearable manually without substantial effort, try to stay above a diameter of 3 mm.

The loose powder in small holes tends to compact, thus a substantial amount of force is required to push it out. While it is still possible to clear those holes out, you will have to use some specialized rigid tools or drills.

Larger holes up from a diameter of 5 mm can, to a certain height, be easily cleaned out using only compressed air.

Try to **avoid designing blind holes**. If the hole is enclosed on one side, it is substantially more difficult to clear as the loose powder cannot be pushed out.



clearable with compressed air

clearable manually

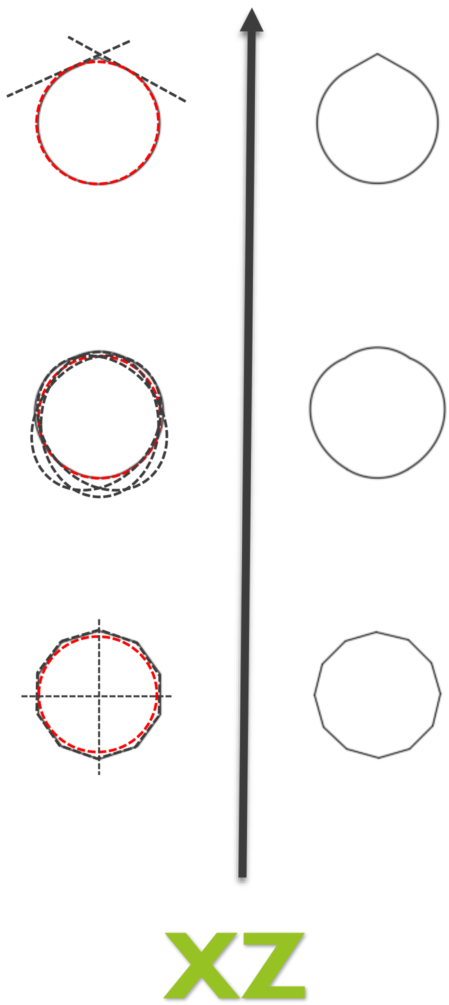
not clearable manually

If you are designing larger parts with smaller holes (< 3 mm), be aware that it may take some effort to clean the holes out. Avoid blind holes if possible.

Holes with a diameter greater than 3 mm can usually be cleared without any major effort.

HORIZONTAL HOLE CORRECTION

If you are printing holes oriented in the XY direction, there are certain design measures you can take to improve the shape of the holes.



Teardrop shape
Creating a teardrop shape with two tangent lines at the top part of the hole will mitigate sagging by a great margin, however a slight cut at the top part of the hole will be visible.

Elliptical shape
Overlaying the top part of the hole with at least 3 symmetrical ellipses has shown to improve roundness and reduce sagging.

Polygonal shape
A simple measure that has shown to be very effective is to overlay the hole with a polygon. Here it is essential to place the vertex of the polygon on the vertical axis of the hole or else it will not work properly.

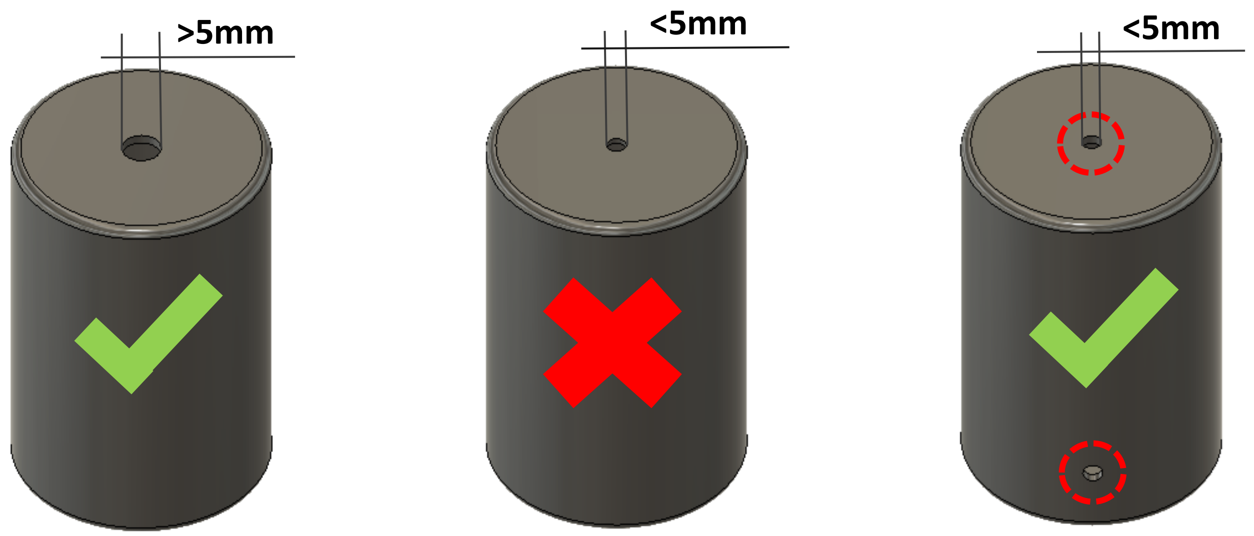
The above-mentioned solutions will not make the hole perfect, however they can improve the roundness and reduce sagging of the upper part of the hole.

As this is a more experimental approach, you will have to test the shapes and dimensions to find the best fit for your design.

ESCAPE HOLES

If you wish to print enclosed cavities, you need to include escape holes to get rid of the excess powder.

A single escape hole will work well up to a diameter of 5 mm to sufficiently clean the cavity of leftover powder if using compressed air.



Be cautious if using compressed air to clean a cavity with only one escape hole of a small diameter (< 5 mm), the pressure may cause the part to burst!

Therefore, if small escape holes are required, at least two are recommended to release the pressure and prevent bursting.

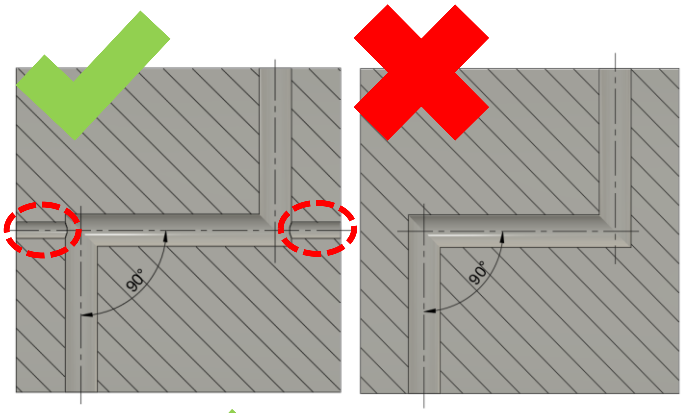
CHANNELS

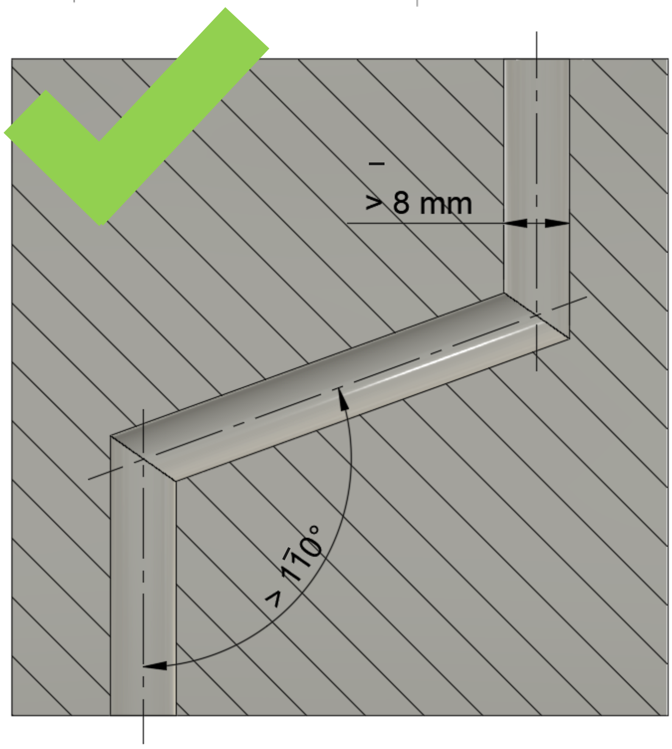
Channels without escape holes will work to a certain extent. However, they are especially challenging to depowder so some adjustments to the design are required.

To be able to remove the loose powder from the channel, it is necessary to design the hidden interconnection between the entrance and exit holes at an angle.

Channels that are connected at a right angle will be challenging to depowder without the use of specialized tools, regardless of the diameter.

Depending on the hole diameter, you may need to adjust the angle of the interconnection – smaller holes will require a bigger angle, bigger holes will work with smaller angles.





The smallest angle and diameter of the interconnection between the entry and exit holes that is possible to depowder without the use of escape holes is at least 110° with a diameter of at least 8 mm.

Generally, angles greater than 110° will be substantially easier to depowder.

EMBOSSED FEATURES

For good readability, the lettering should **not be smaller than 4 mm in height**.

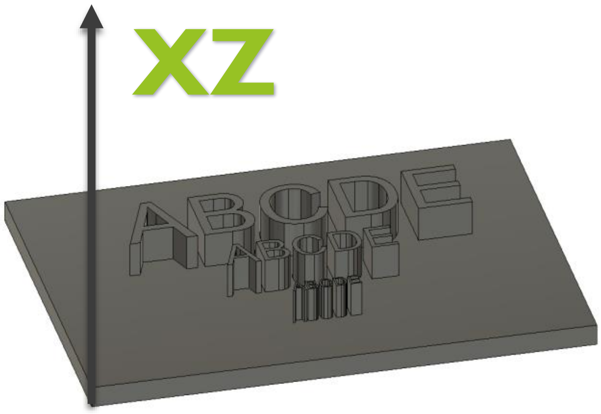
A protrusion height greater than 1 mm will create distinct shadows around the lettering, thus improving the readability.

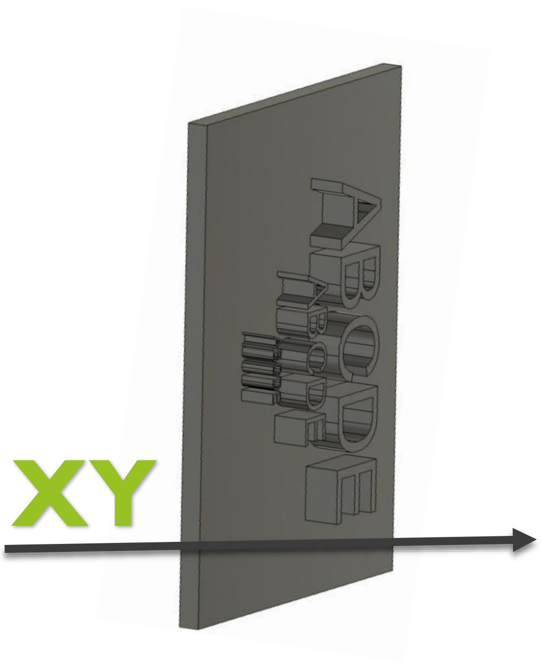
If possible, **use a bold font**. This can improve the readability massively, especially for small lettering.

A **protrusion height greater than 2 mm** makes the embossed features more fragile.

Generally, embossed features printed facing the **XZ direction** will deliver a crisper and **high-resolution finish**.

However, features printed in the **XY direction** seem to be **more robust.**





The sweet spot for embossed features and lettering is a protrusion height between 1 to 2 mm and a lettering height of at least 4mm.

ENGRAVED FEATURES

For good readability, the lettering should **not be smaller than 4mm in height.**

An engraving depth greater than 0.75 mm will create distinct shadows in the lettering, thus improving the readability.

If possible, **use a bold font**. This can improve the readability massively, especially for small lettering.

An **engraving depth greater than 2 mm** in depth causes the lettering to collapse inwards, making it look **softer.**

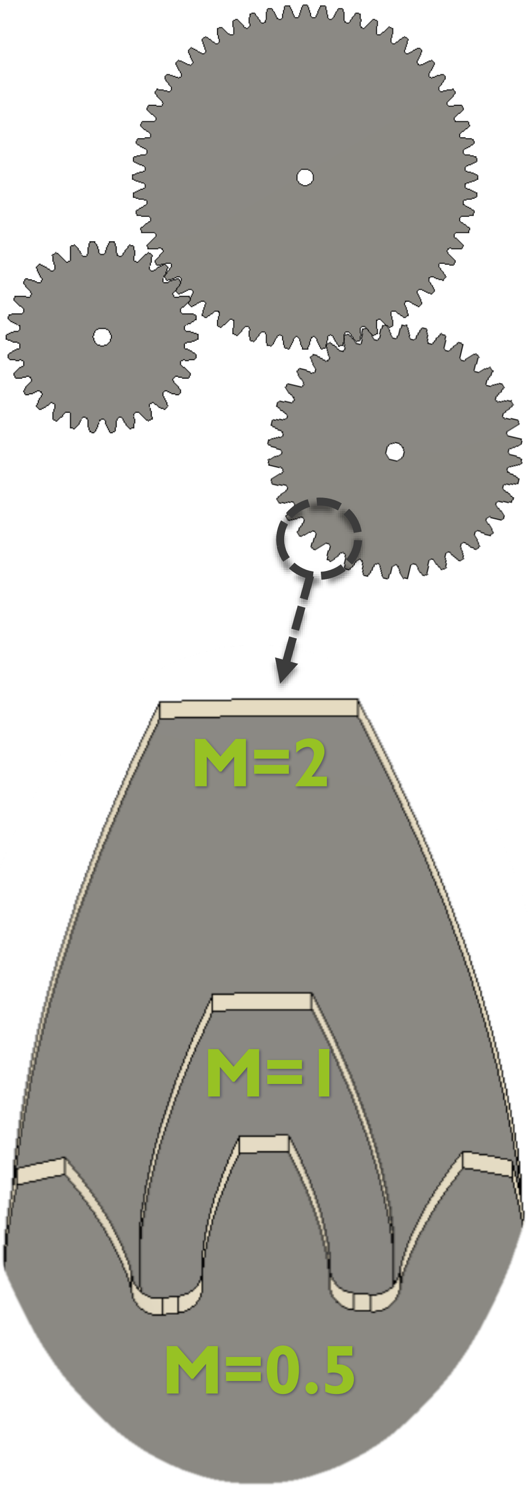
Generally, embossed features printed facing the **XZ direction will** deliver a crisper and **high-resolution finish.**

However, printing the features facing the **XY direction** seems to **preserve very fine details better** compared to the XZ direction.

The sweet spot for engraved features is an engraving depth between 0,75 to 2 mm and a lettering height of at least 4mm.

SPUR GEARS

Spur gears are usually classified via the Module (M), which equals to the pitch circle diameter divided by the number of teeth.



The smallest printable gear module has shown to be 0.5 mm – anything smaller than M = 0.5 will result in poorly resolved teeth and thus a non-functioning gear.

As for the orientation, it is recommended to place the part horizontally to ensure an even teeth resolution and uniform diameter.

There are three main points you should keep in mind when designing spur gear mechanics:

Only gears with the same module will mesh correctly.

The gears will mesh correctly, only if mounted on parallel shafts.

Interference can occur between the gears if the number of the teeth on the smaller of the two gears is less than the required minimum - this can be solved with undercuts on the bigger gear teeth, however it will decrease the strength of the teeth.

THREADS

Threads will generally resolve very well up until a pitch of 1 mm.

Threads with a pitch between 1 mm and 0.75 mm will resolve but, they will not function optimally. A pitch below 0.75 mm is not recommended due to the resulting low resolution of the thread.

Thus, any standard ISO metric thread should work as intended up to the size of M6x1.

However, the optimal pitch of threads has shown to be at least 1.5 mm. Therefore, the recommended size for threads, bolts and screws is at least M10x1.5.

To ensure a uniformly resolved thread and a constant diameter, it is necessary to print the part facing the XZ direction.

It is not recommended to print threads with a pitch smaller than 1 mm.

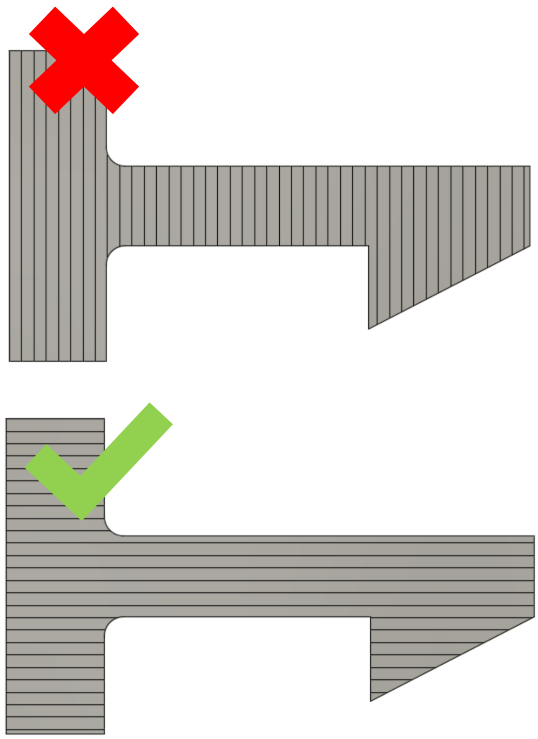
The optimal minimal size for threads is a pitch of 1.5mm (M10x1.5).

It is recommended to print the thread oriented in the XZ direction for optimal results.

CANTILEVER SNAP FITS

Cantilever snap fits are a more complex geometry and require a deeper understanding of plastic construction and assembly. To be successful with your design some amount of calculation or trial and error will be necessary.

Depending on the design of the mechanism, your cantilever snap fit will be either multiple use or permanent – generally both will work well, however for a multiple use snap fit the choice of material matters. Less ductile materials such as PPP will break substantially faster, thus cannot be recommended for the use in multiple use snap fits. TPP materials are more ductile and thus better suited for snap fits in general.

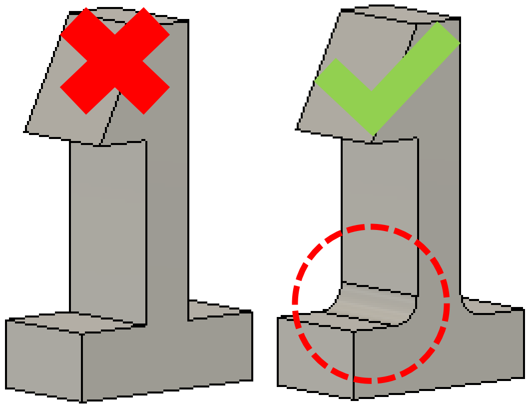


Although the parts tend to behave nearly isotropic after the curing process, anisotropy may occur to a certain degree in delicate structures. For this reason, it is recommended to print the cantilever facing the XY direction. Generally, you should orient the cantilever in such a way, that the mechanical strain is perpendicular to the layer build-up of the cantilever. This will result in a more strainable and durable cantilever snap fit.

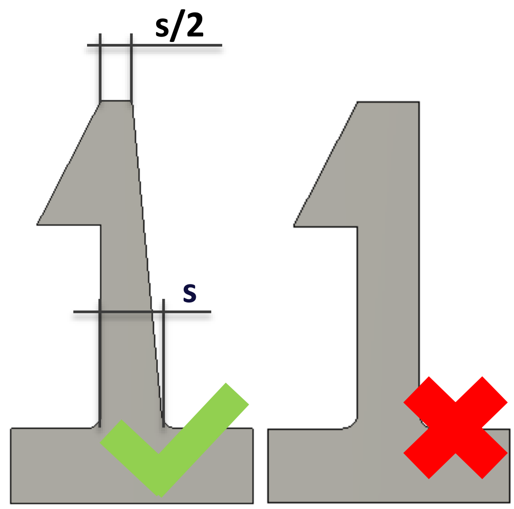
CANTILEVER SNAP FITS

There are certain aspects that you should consider when designing cantilever snap fits to be successful:

Always fillet the base
A fillet at the base of the cantilever is essential to ensure stability and longevity as the maximum stress of a cantilever snap fit usually occurs at the transition between the cantilever and the part surface. As a rule of thumb, you should include a **fillet radius of at least half the size of the width** of the cantilever.

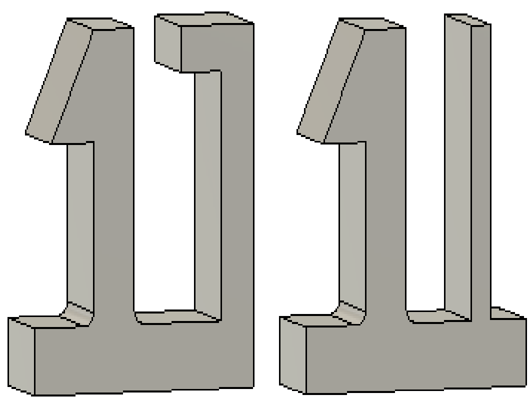


Shape of the Cantilever
There are various shapes of cantilevers, ranging from a simple rectangular shape to trapezoids and complex irregular cross sections. A design feature that has been proven as effective is, to **taper down the thickness** of the cantilever from the base to the top. This will lead to a more uniform stress distribution throughout the cantilever.



Avoid stress outside of the assembly
Make sure that the cantilever is **not under stress after the assembly** to avoid material fatigue.

Overextension protection
It is recommended to protect the cantilever from overextending, especially when working with delicate structures. This can be achieved with ribs placed behind the cantilever or simply building in constrains in your design.

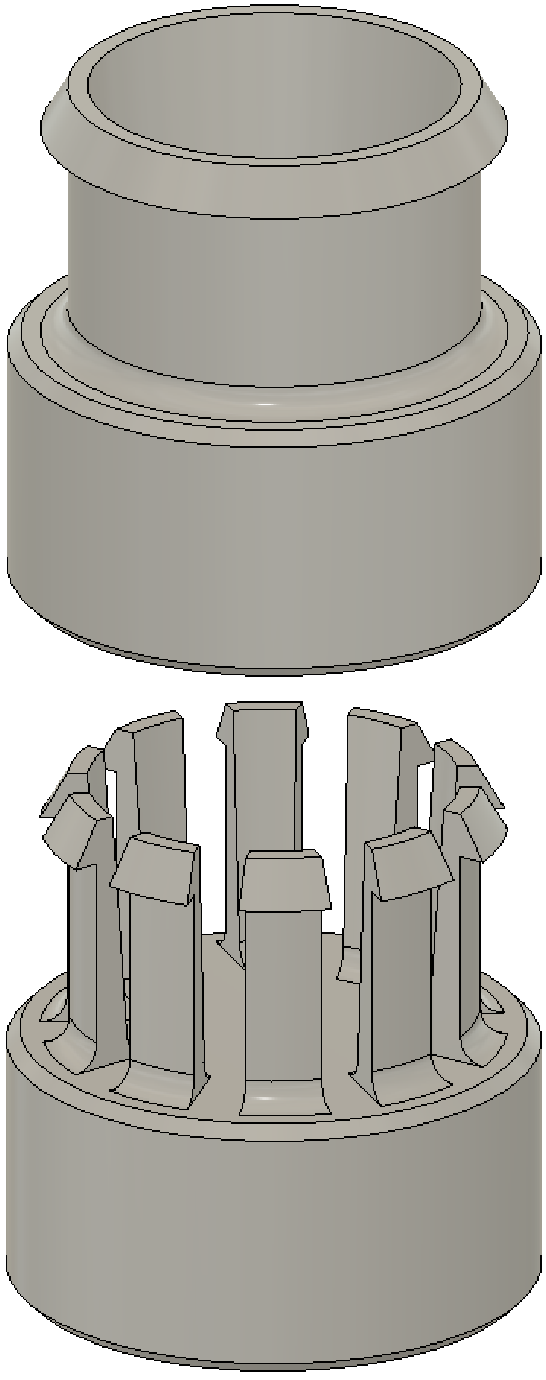


ANNULAR SNAP FITS

Before designing annular snap fits, it is recommended to calculate important characteristics such as, the deflection, deflection force and mating force. This will provide you with a good estimate about the suitability of the material for the use in annular snap fits. Typical annular snap-fits that utilize hoop-strain will not work with rigid materials.

A suitable alternative that has shown good results are segmented annular snap fits that work on the principle of a flexural beam, same as typical cantilever snap fits. In this case all the points mentioned in the chapter cantilever snap fits apply here as well.

Similar to cantilever snap fits, it is recommended to print the annular snap fit facing horizontally or rather in such a way, that the mechanical strain is perpendicular to the layer build-up.



Annular snap fits are not suited for very rigid materials. A viable alternative is segmented annular snap fits that work on the principle of a flexural beam.

BALL JOINTS

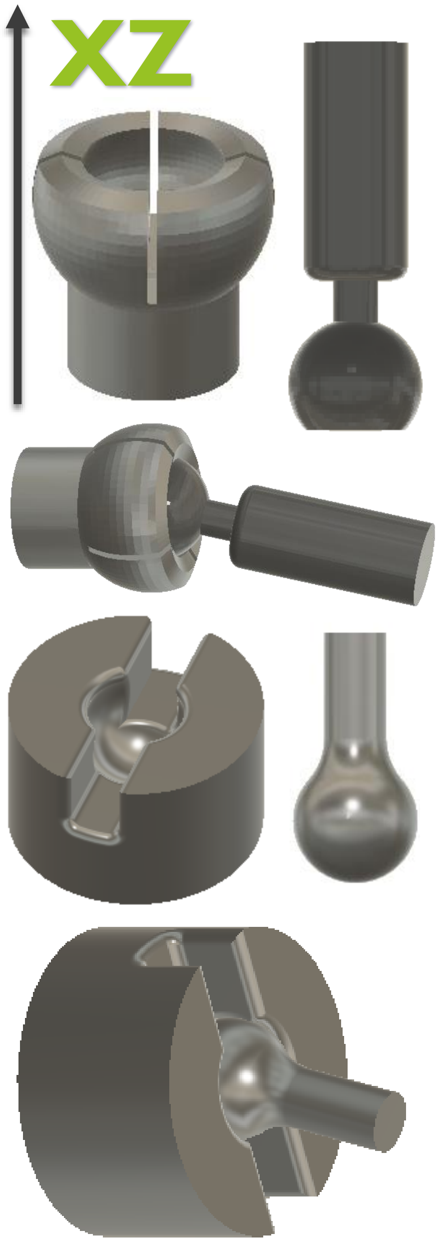
Ball joints will generally work very well however, they **must be printed facing the XZ direction** to ensure a perfect fit.

If printed facing the **XY direction**, the socket and ball will **most likely not fit** together as intended.

Ball joints placed in a more **massive socket** will be **difficult to disassemble** due to the rigid nature of the material.

If you want to be able to disassemble your ball joint, a thinner socket is recommended. **Separations in the socket** and a slight **cut at the end of the ball** will also improve the **separability**.

To ensure good stability, the **shaft** attached to the ball should have a **diameter** of at least **6 mm**.



It can be tricky to get the dimensions between the ball and the socket right at first - always test your design before printing a high quantity.

If you are having trouble with the fit of the ball joint, try printing multiple iterations of the ball with varying diameters to find the correct dimensions.

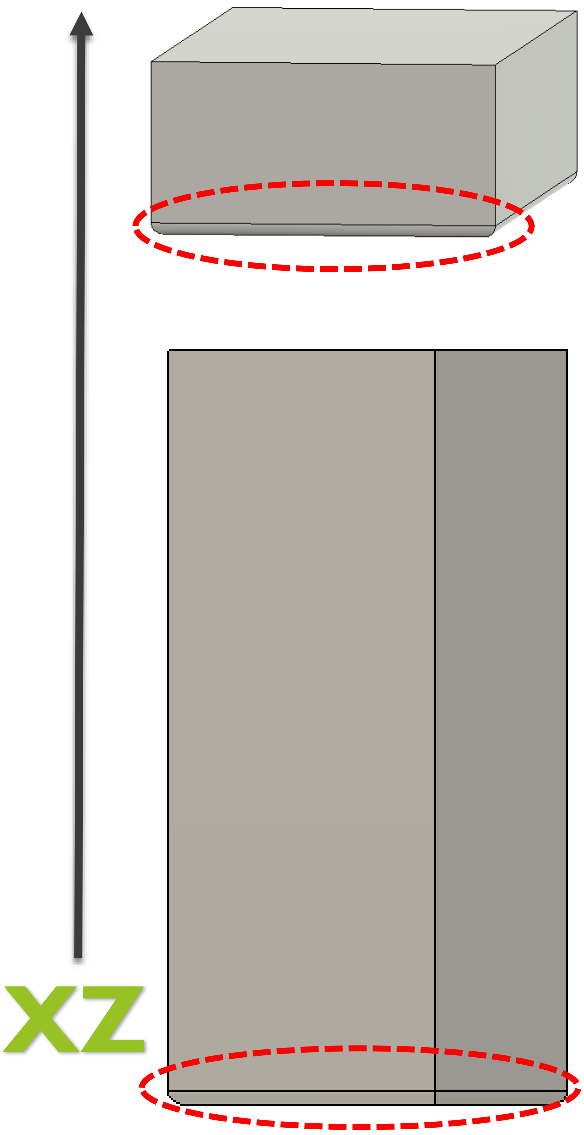
SURFACE FINISH

The orientation of the part during the printing process has an influence on the resulting surface finish. The reason for this phenomenon is the SLS process itself – the downwards-facing side does not have a solid lower layer beneath itself, thus the surrounding loose powder is exposed to more thermal energy.

This effect is not overly significant for stability. However, it is noticeable upon close inspection and should be considered if the optical properties and haptics matter.

The downwards-facing side of the part, also called downskin, will end up with a softer finish and the edges tend to round slightly.

The upwards-facing side of the part, also called upskin, will be crisper and sharper. Finer details tend to be resolve better and more accurately.



Generally, it is recommended to orient important features that need high accuracy and good resolution facing upwards.

Features oriented downwards will be less sharp and the edges tend to round up.

CONCLUSION

Selective Laser Sintering is a powerful and versatile technology. However, like any other manufacturing process, it has its peculiarities that you must understand in order to achieve consistent results. The design guidelines should provide you with an outlook on what is doable with our materials as well as a basic understanding of SLS itself.

In conclusion, you should keep the following points in mind when designing parts for SLS 3D-printing:

- Optimize your design for the SLS technology. Reusing designs that were meant for different manufacturing processes such as injection molding will often not work as intended with SLS 3D-printing.

- Always consider all steps of the manufacturing process. All steps, beginning from the orientation in the part bed, depowdering and curing have an impact on the part quality.

- If possible, test your designs before printing a high quantity. This can save you a lot of time and material.

- Printing parameters such as the hatch distance and laser power have a major influence on the outcome of your part. Every machine has its peculiarities, parameters working on one printer may not necessarily deliver the same outcome on another. Thus, all dimensions mentioned in the design guidelines are not necessarily set in stone but should rather be viewed as a general approximation and to show what is doable. We are steadily expanding the classification of our materials for various printers. In the meantime, you are welcome to reach out to our team at TIGITAL 3D-Set if you require any support regarding our materials.

INFORMATION TO THE USER - DISCLAIMER

TIGER's verbal and written recommendations for the processing, use and application of our products, including the information, product properties, processing parameters and design recommendations provided in this guideline, are based upon our experience and in accordance with present technological standards. These are given only in order to support the user, are noncommittal and do not constitute any additional commitment or product specification. The user shall be solely responsible for checking and verifying the suitability and/or fitness of our products exemplary processing conditions and design recommendations for the intended use, application and process. Further, the user shall be solely responsible for the appropriate, safe and legally compliant use, processing, handling and application of our products as well as for any related 3D printed parts.

TO THE EXTENT PERMITTED BY APPLICABLE LAW, TIGER, ITS EMPLOYEES, BOARDS, OWNERS, AGENTS, SUBCONTRACTORS OR AFFILIATES CAN NOT BE MADE LIABLE FOR ANY DAMAGES, WHETHER CAUSED DIRECTLY OR INDIRECTLY BY OR BEING RELATED TO THIS GUIDELINE, OR FOR ANY RELATED LOSS OF PROFIT, LOSS OF GOODWILL, OR ANY OTHER CONSEQUENTIAL DAMAGES.