The Ultimate HP Jet Fusion (MJF) Design Guidelines

This page attempts to be the ultimate HP Jet Fusion 3D printer design resource. This content is straight from HP white papers but has never before been formatted outside of a pdf. This page makes that content much more accessible and searchable to Rapidmade customers on desktop and particularly mobile. As one of the largest power users of HP technology it is imperative to us that you have all the information to get the most out of this process. We look forward to helping you make the best 3D parts on the planet!

Table of Contents

Best MJF Design Practices

Introduction

As with other 3D printing technologies, there is a set of recommendations to follow when designing for HP Multi Jet Fusion technology to ensure parts and features are printed to specification.

Recommended Wall Thickness

In general, the minimum recommended wall thickness is 0.3 mm for short walls oriented in the XY plane, and 0.5 mm for short walls oriented in the Z direction.
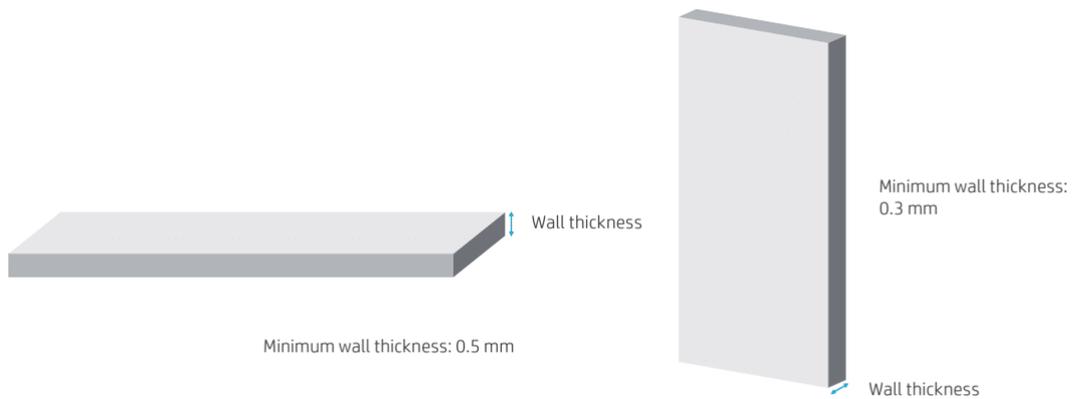
Figure 1. Minimum wall thickness

Cantilevers

When printing a cantilever, the minimum wall thickness depends on the aspect ratio, which is the length divided by the width.

For a cantilever with a width of less than 1 mm, the aspect ratio should be less than 1. There are no specific recommendations for widths of 1 mm or larger. For parts with a high aspect ratio, it is recommended to increase the wall thickness or to add ribs or fillets to reinforce the part.
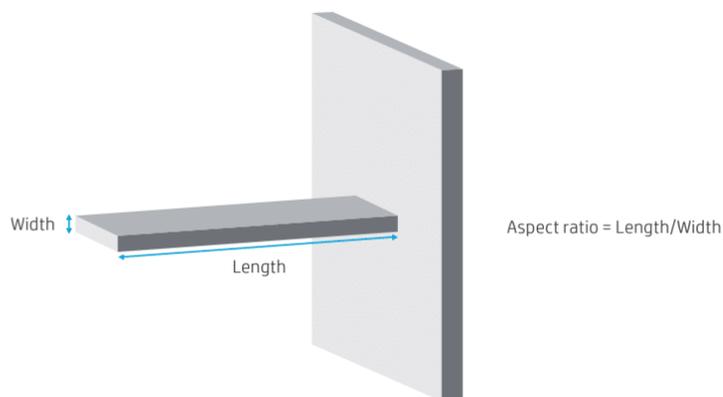

Figure 2. Cantilevers

Connecting Parts

Sometimes a pair of printed parts need to fit together to form the final application. To ensure correct assembly, the minimum gap between the interface areas of these parts should be at least 0.4 mm (±0.2 mm of tolerance for each part).
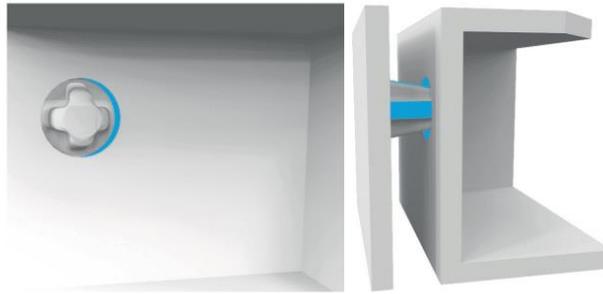
Figure 3. Minimum gap between connecting parts

Moving Parts

As a general rule, spacing and clearance between faces of printed as assemblies should be a minimum of 0.7 mm.

Parts with walls with a minimum thickness of 30 mm should have a larger gap between each side to ensure proper performance.

For parts with walls that are thinner than 3 mm, the clearance between parts printed as assemblies can be as low as 0.3 mm, but this fully depends on the design, and iterations with the manufacturer may be necessary to ensure quality performance.
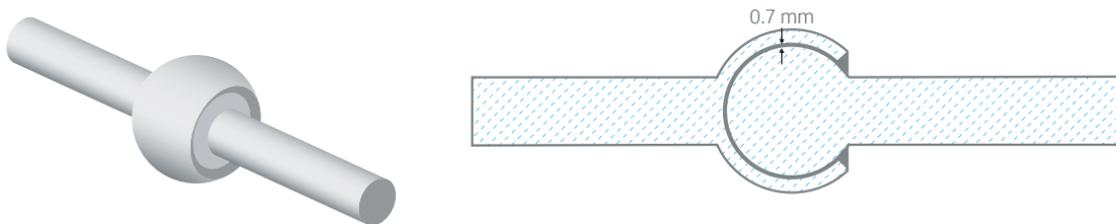

0.7 mm

Figure 4. Minimum gap between moving parts

Thin and Long Parts

Thin and long parts are susceptible to non-uniform cooling, which may cause uneven shrinkage along the printed part, creating a distortion in a certain direction that deviates from the nominal shape.

As a rule of thumb, any part with an aspect ratio—length vs. width—higher than 10:1, or any part with an abrupt change in its cross-section or a predominantly long and thin curved segment is susceptible to exhibiting warpage as shown in the image below:
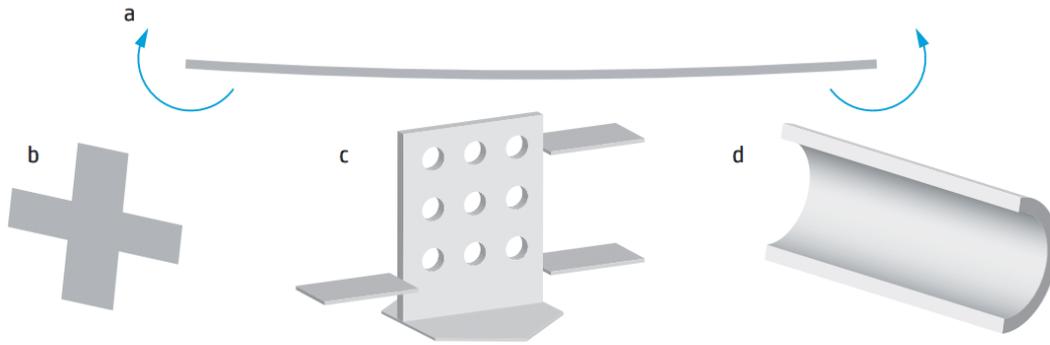
Figure 5. Part categories susceptible to shrinkage-induced warpage (a) include: thin and long parts (b), parts with abrupt changes in cross-sections (c), and thin curved surfaces (d)

To minimize the possibility of this deformation, there are several recommendations to keep in mind when designing the part:
• Increase the thickness of long walls to reduce their aspect ratios.
• Avoid ridges and ribs on large, flat areas.
• Re-design parts with high potential stresses and smoothen their cross-section transitions.
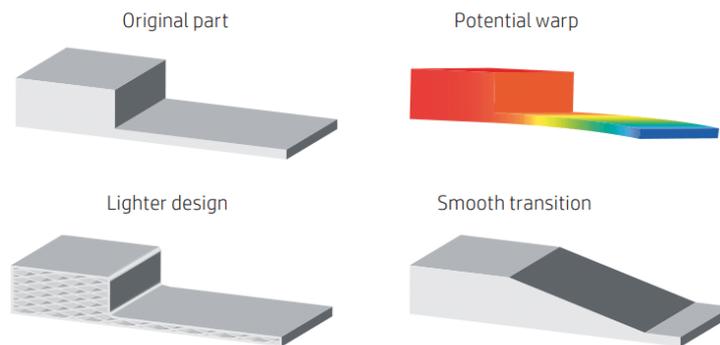• Lighten the parts by hollowing them or by adding internal lattices.

Figure 6. Warpage mitigation strategies

3D Print Design optimization strategies: Solid part or structural fill

HP Multi Jet Fusion technology allows for the printing of topology-optimized, generative designs or even small lattice structures. This kind of design allows for the creation of thinner sections, which accumulate and re-radiate less heat, improving the dimensional accuracy and general look and feel of the parts.

It also helps to reduce the weight of the part, the quantity of material, and the fluid agent used compared with fully solid designs, which not only reduces the cost of the part but also helps reduce the operating cost in applications that are very weight-sensitive.

Hollow Parts

This design optimization strategy involves hollowing the model through an automatic process. (Professional software such as SolidWorks, Materialise Magics with Materialise Build Processor for HP Multi Jet Fusion technology, and Autodesk® Netfabb® have this built in.)

The minimum recommended wall thickness is 2 mm, but higher mechanical properties are achieved with thicker walls. The optimum choice is dependent upon the application.

Once the model has been printed, drain holes can be implemented in the hollow part to remove the trapped unfused powder. Otherwise, trapped unfused powder can remain within the part, which results in heavier and more resistant parts compared with the fully hollow option. While the part is still light, it is weaker than the non-hollowed version. The difference in weight stems from the different densities of fused and unfused material.
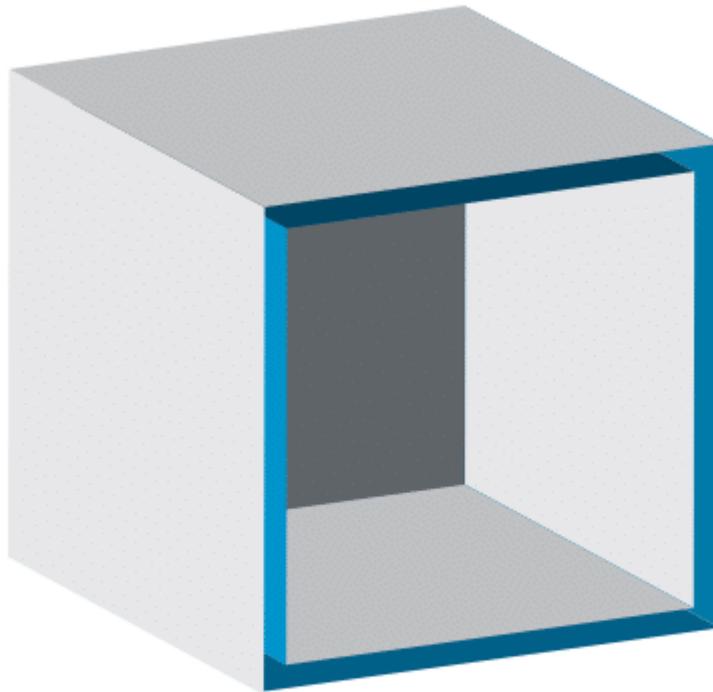

Figure 7. Example of hollow part

Leaving the powder trapped within a part also saves post-processing time since powder extraction is not required.

Lattice Structures

This design optimization strategy involves hollowing a part and replacing the internal solid mass with a lattice structure that provides mechanical integrity via the collective action of many rigid cells while still noticeably reducing the part's mass and cost.

This re-design is also a fast process that can be automated with professional software such as Materialise Magics or nTopology.
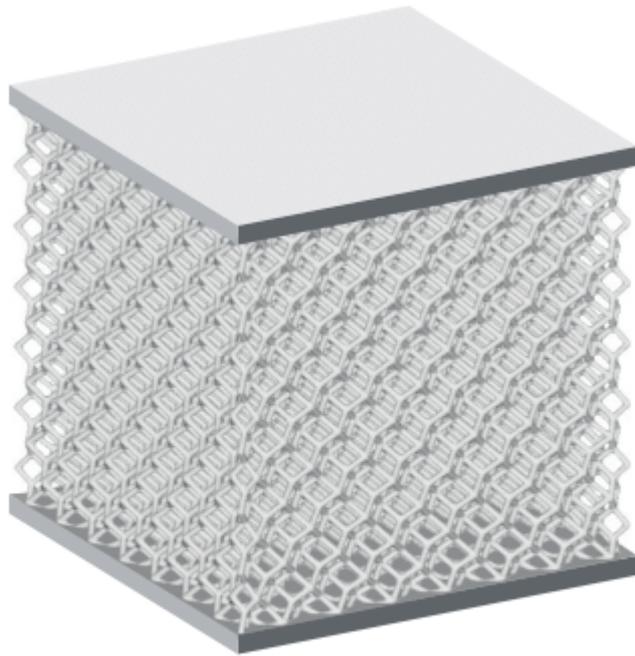
Figure 8. Example of lattice structure

Topology Optimization

Topology optimization is a finite element method (FEM)–based process that finds the best distribution of material given an optimization goal and a set of constraints. Typical optimization goals are mass reduction and creating specific mechanical properties. This process requires the designer to know the part's function and load distribution in depth but provides the most optimized method of reducing weight and cost from the original design.

*HP part optimized
by Autodesk with Netfabb*

Figure 9. Example of topology optimization

Design for 3D Print Accuracy

Introduction

To avoid issues with parts and to achieve maximum accuracy when designing with HP Multi Jet Fusion (MJF) technology, there are certain specifications to bear in mind.

Dimensional Accuracy

When designing parts with HP Multi Jet Fusion technology, it is possible to achieve accuracy values of IT Grade 13, with Cpk values that rival those of plastic Injection Molding.

Minimum Specifications for MJF Parts

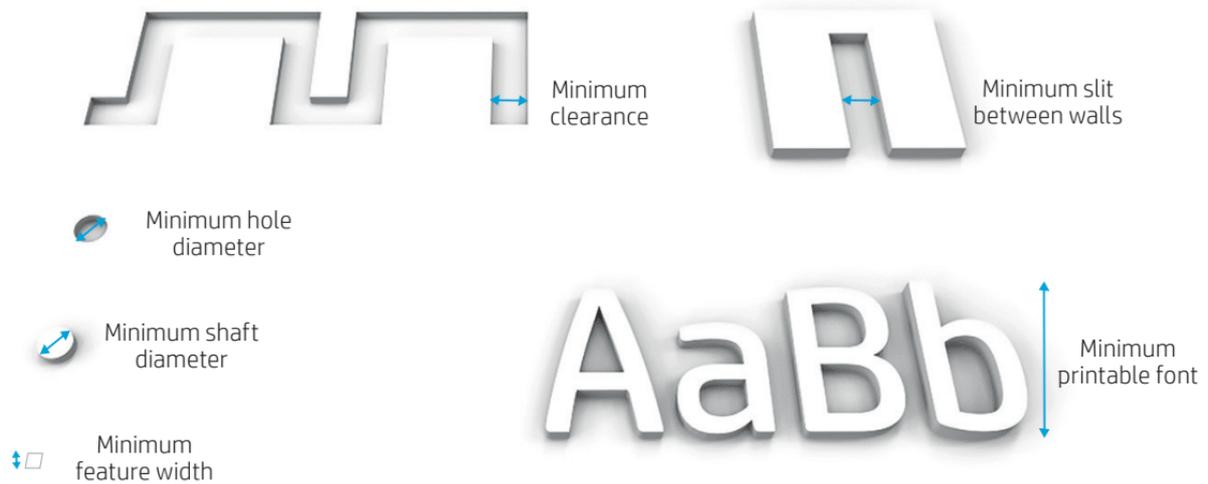|	
Minimum hole diameter at a thickness of 1 mm	0.5 mm
Minimum shaft diameter at a height of 10 mm	0.5 mm
Minimum printable font size for embossed or debossed letters or numbers	6 pt
Minimum printable features or details (width)	0.1 mm
Minimum clearance at thickness of 1 mm	0.5 mm
Minimum slit between walls/embossed details	0.5 mm

Figure 1. Minimum specification for parts

Embossed and Engraved Details

HP Multi Jet Fusion technology allows users to print embossed and engraved details such as letters and drawings with very high resolutions and definitions.

For the best possible output, any text, number, or drawing included in a part should have a depth or height of at least 1 mm.
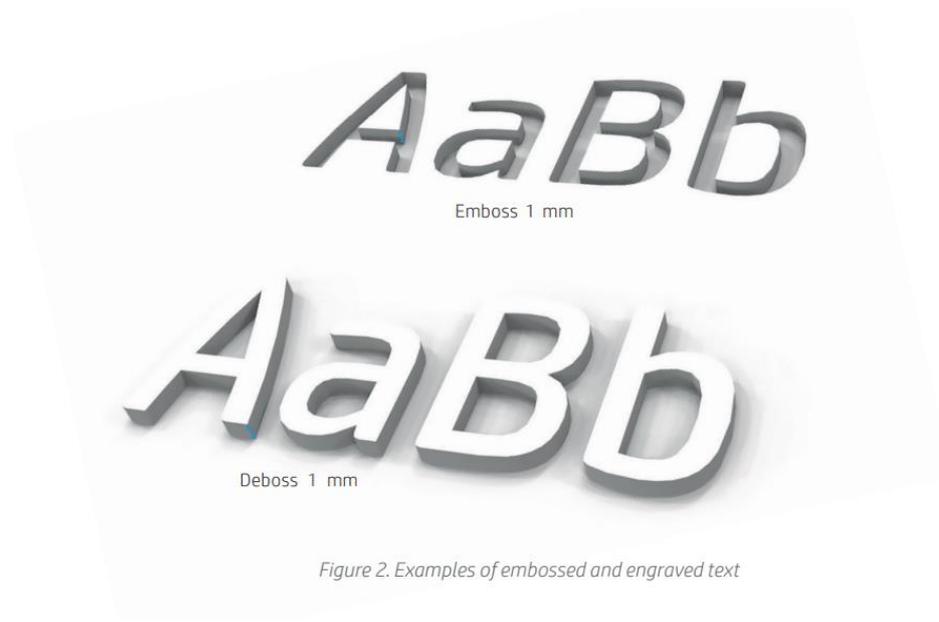

Figure 2. Examples of embossed and engraved text

3D Print Accuracy Guidelines

• When possible, place small features with critical dimensions—such as pins, holes, and raised texts—in the same plane.
• Design parts with a smooth cross-section transition.
• When possible, design lighter parts by hollowing them or adding internal lattices.
• Avoid long, thin, flat parts with an aspect ratio—length vs. width—higher than 10:1.
• Avoid design parts with predominantly long and thin curved segments.
• Avoid ridges and ribs on large, flat areas.

Design for Aesthetics / Best Surface Finish on your 3D Prints

Introduction

To print parts with optimal appearance and material properties, there are certain specifications to bear in mind.

3D Print Layer Lines - "the Stair-Stepping Effect"

All layer-by-layer manufacturing technologies require a discretization of their Z dimensions according to the layer thickness. The visibility of these layers depends mainly on their thicknesses and printing angles.

HP Multi Jet Fusion (MJF) technology uses layers of only 80 µm (0.080 mm), which are difficult to see with the naked eye in most situations. However, for small angles in the part, layered steps could become visible.

Thus, when designing parts with protruding features, it is recommended to keep angles above 20° between big, flat areas and the XY plane if they will be facing upward. Surfaces that face downward are typically exempt from stepping as long as they are oriented and avoid angles less than 5° to 10°.

These values, however, are general indications and ultimately depend on the application. For optimum results, the best solution is to try several options and choose the one that yields the better look and feel.
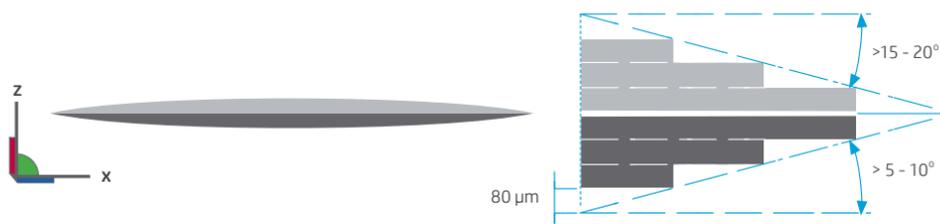

Figure 1: Stair-stepping effect

MJF Aesthetics Guidelines

• When possible, place small features with critical dimensions—such as pins, holes, and raised texts—in the same plane, taking into account that areas printed facing downward would have a better look and feel than those that face upward.
• Design parts with a smooth cross-section transition.

• When possible, add internal lattices or hollow the parts to achieve a lighter design.
• Avoid long, thin, flat parts with an aspect ratio—length vs. width—higher than 10:1.
• Avoid designing parts with a predominantly long and thin curved segment.
• Avoid ridges and ribs on large, flat areas.

Design for Cleaning Jet Fusion Parts

Introduction

Ease of cleaning is one of the advantages of HP Multi Jet Fusion technology compared with other 3D printing technologies.

However, in terms of 3D printing production, designers should take into account several recommendations in order to facilitate
the cleaning process and minimize the cost once the part is printed.

Drain Holes

When printing hollow parts, add at least two drain holes on opposite faces of the part for efficient powder removal, which is critical to obtain the largest weight reduction. The minimum recommended diameter of the drain holes is 5 mm.

Lattice Structures

Unfused powder can be difficult to remove from a part through drain holes when a part has a lattice structure inside. Therefore, it is recommended to leave the powder trapped inside or to leave the lattice partially open. The minimum gap recommended in a lattice structure to ensure that the material inside the part can be removed is 5 mm.
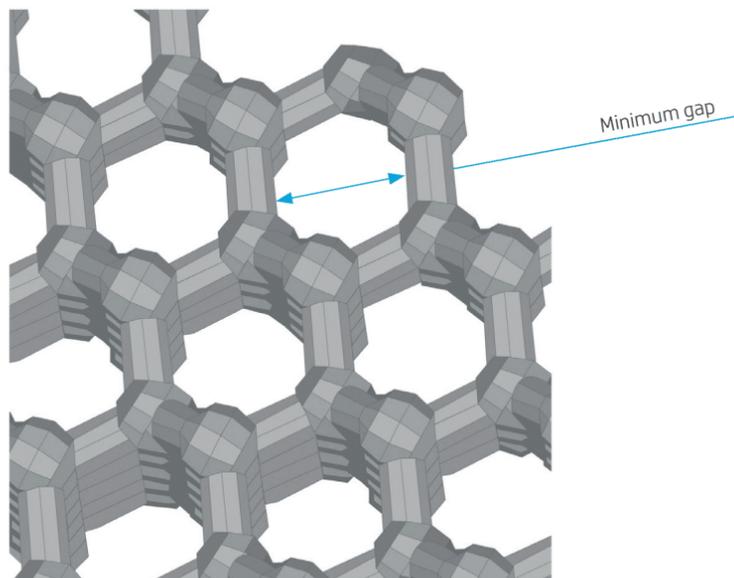

Figure 1. Minimum gap for lattice structures

Ducts

To remove material from narrow ducts, design and print a strip or a chain through the duct. When the part has been printed, the chain can be pulled out to dislodge most of the material.

Any remaining material can be removed through the normal cleaning process.

Figure 2. Duct cleaning

For ducts narrower than 5 mm, clean the inside with a flexible screw once the part has been printed. To improve the flexible screw cleaning performance, it can be attached to a drill.

Figure 3. Flexible screw

Dimensional Tolerancing 3D Prints

Introduction

HP Multi Jet Fusion technology allows for the designing and printing of parts that can be assembled between them or to other manufactured parts, such as metal parts, to create final products and functional assemblies. The parts can be joined by union joints such as self-tapping screws, threaded inserts, or snap-fits.

It is important to consider tolerances at an early stage of the product development process and to design every part involved in a final product or functional assembly taking into account the permissible range of variation in dimensions to ensure that it fits suitably and works according to the design intent.

Depending on how the parts must interact to create a final product or achieve the assembly's functional needs, the required tolerances will be tighter or wider, which will require the most capable manufacturing process to produce the part with suitable accuracy.

International Tolerance (IT) Grades

Designing a part often involves the use of the International Tolerance Grades defined in ISO 286/ANSI B4.2-1978, which provide a standardized reference for typical manufacturing process capability in terms of tolerance accuracy for a given dimension.

The most common manufacturing processes have an associated IT Grade that specifies their capability to provide accurate parts, as shown in the image below:

Figure 1: International Tolerance (IT) Grades

Each IT Grade establishes the allowable tolerance limits for a given dimension. As shown in the following table, a smaller IT Grade provides tighter tolerances:

Standard tolerance grades	Values of standard tolerance (mm)								Nominal size (mm)
	from: 1 to: 3	3 6	6 10	10 18	18 30	30 50	50 80	80 120	
1	0.0015	0.0015	0.0015	0.0015	0.0015	0.002	0.002	0.003	Measuring tools
2	0.002	0.002	0.002	0.002	0.002	0.003	0.003	0.004	
3	0.003	0.003	0.003	0.003	0.004	0.004	0.005	0.006	
4	0.004	0.004	0.004	0.005	0.006	0.007	0.008	0.010	
5	0.005	0.005	0.006	0.008	0.009	0.011	0.013	0.015	Engineering fits, bearings, machining processes (grinding, turning)
6	0.007	0.008	0.009	0.011	0.013	0.016	0.019	0.022	
7	0.009	0.012	0.015	0.018	0.021	0.025	0.030	0.035	
8	0.014	0.018	0.022	0.027	0.033	0.039	0.046	0.054	
9	0.025	0.030	0.036	0.043	0.052	0.062	0.074	0.087	
10	0.040	0.048	0.058	0.070	0.084	0.100	0.120	0.140	
11	0.060	0.075	0.090	0.110	0.130	0.160	0.190	0.220	
12	0.090	0.120	0.150	0.180	0.210	0.250	0.300	0.350	Large manufacturing, die casting, stamping, sand casting
13	0.140	0.180	0.220	0.270	0.330	0.390	0.460	0.540	
14	0.250	0.300	0.360	0.430	0.520	0.620	0.740	0.870	
15	0.400	0.480	0.580	0.700	0.840	1.000	1.200	1.400	
16	0.600	0.750	0.900	1.100	1.300	1.600	1.900	2.200	
17	0.900	1.200	1.500	1.800	2.100	2.500	3.000	3.500	
18	1.400	1.800	2.200	2.700	3.300	3.900	4.600	5.400	

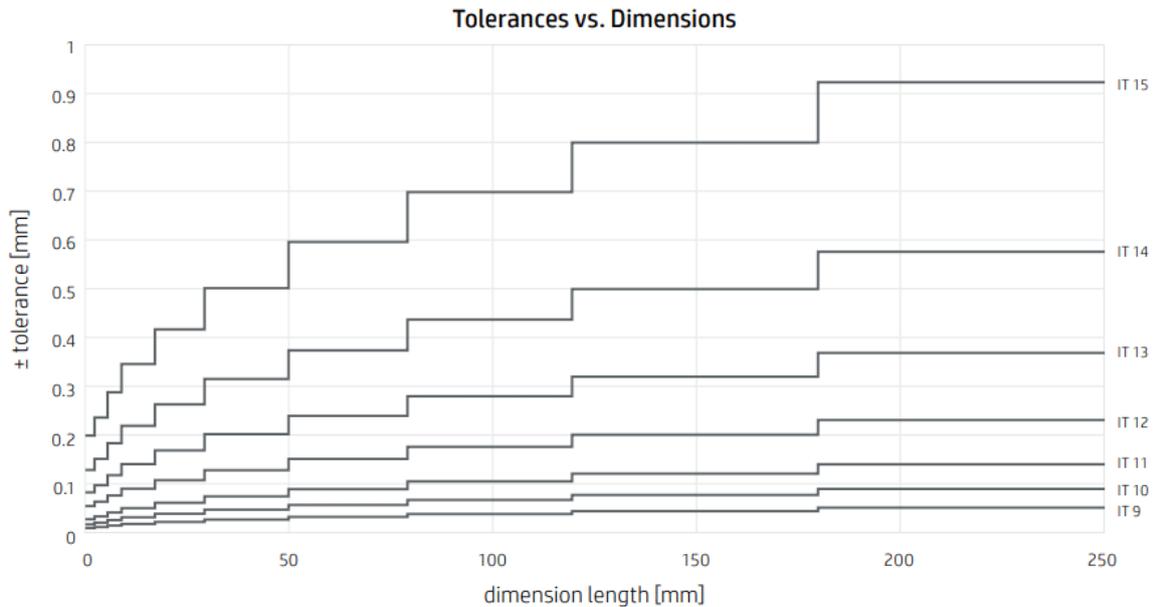Table 1: Standard tolerance grades

Figure 2: Graph of International Tolerance (IT) Grades

Process Capability

Process capability determines whether a process meets a specification. The process capability index or process capability ratio or Cpk is a statistical measure of process capability. It quantifies the ability of a process to produce output within specification limits.

When talking about a dimensional specification, the Cpk measures the statistical probability that a certain process produces a dimension within its tolerance range. The higher the Cpk value the better, meaning that more measurements will be within its tolerance range.

For a process to be capable, it needs to be both **repeatable and accurate.**

Repeatability is how close multiple measurements are to each other (also called precision).

Accuracy is how close a measurement value is to the specified nominal.

The capability of a process is then a function of two parameters:
• How repeatable it is compared to the width of the specification limits, measured by the Cp
• How accurate it is, measured by the **bias**

$$Capability = C_{pk} = C_p * (1-2*bias)$$

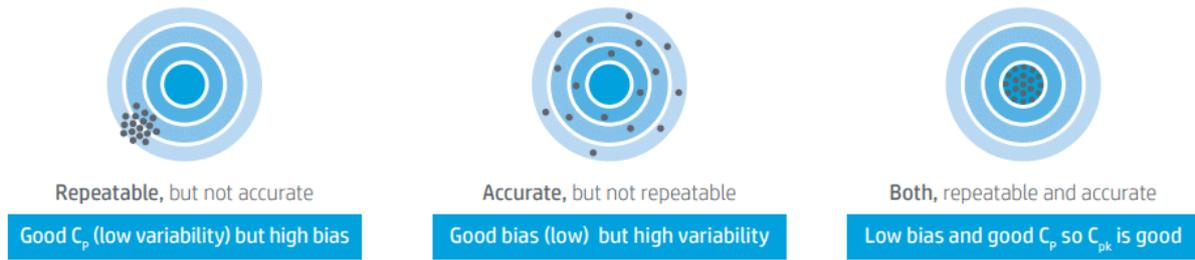| Repeatable, but not accurate	Accurate, but not repeatable	Both, repeatable and accurate
Good C_p (low variability) but high bias	Good bias (low) but high variability	Low bias and good C_p so C_{pk} is good

Figure 3: Relationship between bias and variability

This concept only holds meaning for processes that are in a state of statistical control with an output that is approximately normally distributed.

Both conditions happen when dealing with the dimensional quality control of HP MJF–produced parts where the output is the dimensional value of the different geometrical features of a part.

Dimensional quality control processes define an upper specification limit (USL) and lower specification limit (LSL), also called the "tolerance range" of the process. The target of the process is the center of this range, typically the nominal dimension value.

The objective to have a well-controlled dimensional process is to have its normal distributed population of measurements:
• With a variability (calculated as standard deviation) that "fits" in the tolerance range. Cp measures how well the variability fits within the tolerance range.
• With a mean (average) as close as possible to the target. The deviation is measured by the **bias.**

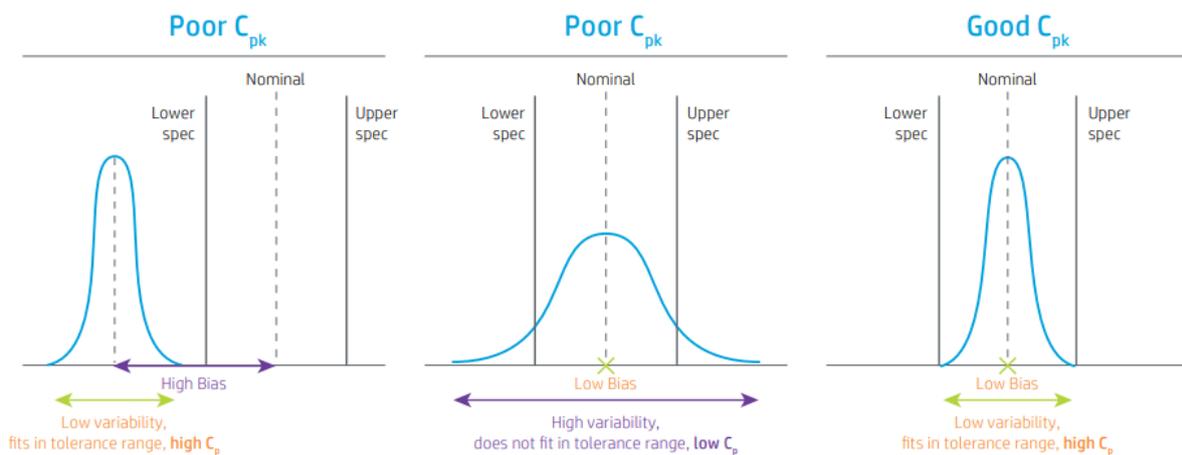Only if both conditions are met, process capability measured by Cpk is considered good:

Figure 4: Process capability C_{pk} scenarios

The mathematical calculation of these parameters is as follows:

$$C_p = \frac{\text{Specification width}}{\text{Process width}} = \frac{(USL - LSL)}{6\sigma}$$

Standard deviation estimates the sigma and quantifies the variability and dispersion of the process. Cp should always be greater than 1.0 for the variability to fit within the tolerance range.

$$C_{pk} = min\left[\frac{[USL - \mu]}{3 \cdot \sigma}, \frac{[\mu - LSL]}{3 \cdot \sigma}\right]$$

The statistical mean estimates the mu (μ).
Therefore:
• Cpk "measures" the distance of the mean to the closer specification limit, which could be the upper or the lower limit.
• Cpk takes into account how centered the process is (Cpk ≤ Cp).
 For a perfectly centered process, Cp = Cpk.
• If Cp > Cpk, it is possible to increase the Cpk by readjusting the mean of the process.

The following table displays the relevant Cpk values and their correlation with process yields:

	C_{pk}	Sigma level	Dimensions within specs (%)	Dimensions out of specs (units per million)	Part yield for a part with 10 dimensions (%)	
100% inspection	0.33	1	68.27	317,300	2.20	
	0.67	2	95.45	45,500	62.77	
	1.00	3	99.73	2,700	97.33	
	1.33	4	99.9937	63	99.94	Desired
Statistical process control	1.50	5	99.99966	3.4	100	
	1.67	6	99.99997	0.6	100	

Table 2: C_{pk} and process yield correlation

For a part to be considered good, all the specified dimensions need to be within tolerances. Therefore, the part yield is a metric that can be calculated as the statistical sum of the single dimension success rate. In the previous table, an example for a part with 10 dimensions is shown in the right column.

For Cpk values below 1, the yield is such that the best quality control method is 100% inspection, and the general fabrication process is to over-produce and send only the parts that meet the tolerance requirements. This is costly but it is a reasonable process, especially for low-volume production.

For Cpk values above 1 (3 sigma), the dimensional success rate and the yield begin to approach each other, and statistical process control starts to become a viable option. This means that after the process has demonstrated that it is statistically and consistently achieving Cpk above 1 for all dimensions, one could move to auditing random parts per each lot of parts.

Generally, a Cpk of 1.33 (4 sigma) is desired to ensure enough of a margin for statistical process control, especially when dealing with multipart complex mechanisms.

HP Jet Fusion 3D Process Control

The HP Jet Fusion 5200 Series 3D Printing Solution has an in-printer feature that provides the capability to apply dimensional profiles. This feature helps streamline the workflow and provide an enhanced experience while helping to achieve manufacturing-level accuracy and repeatability.

By default, the solution comes with a general dimensional profile. Using HP 3D Process Control software, hardware-specific dimensional profiles can be generated and managed to achieve optimized dimensional capability and ensure uniform results across a fleet of printers.

Types of Fit

Introduction

HP Multi Jet Fusion technology allows users to print mating parts to create functional assemblies. When designing mating parts with the suitable tolerance and type of fit, it is important to save time in post-processing and assembly operations.

Fits are used to establish tolerances between inner and outer features of bearings, bushings, shafts, or drilled holes, and are often represented as a shaft and a hole, although they include other parts that are not only cylindrical.

There are two types of fits based on the allowable limits for shaft and hole size:

Clearance fit:

 A clearance fit leaves a space or clearance between mating parts: The hole diameter is larger than the shaft diameter. The shaft can slide and/or rotate in the hole when assembled, requiring no force.
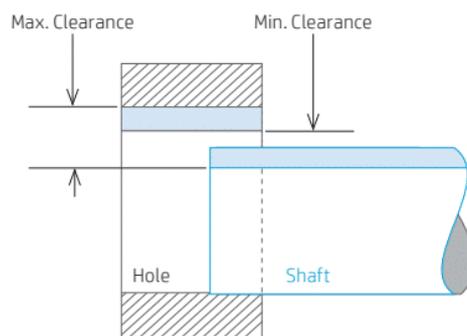
Figure 1: Clearance fit

In this type of fit, the maximum clearance is the difference between the maximum size of the hole and the minimum size of the shaft, while the minimum clearance is the difference between the minimum size of the hole and the maximum size of the shaft.

Interference fit:

In an interference fit the hole diameter is smaller than the shaft diameter. This type of fit does not allow relative motion between mating parts, providing a strong connection and requiring strong force in assembly and disassembly.
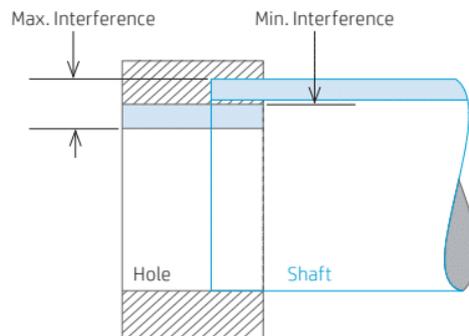

Figure 2: Interference fit

In this type of fit the maximum interference is the difference between the maximum size of the shaft and the minimum size of the hole, while the minimum interference is the difference between the minimum size of the shaft and the maximum size of the hole.

Design Guidelines for MJF Fit

Depending on how the mating parts must fit to achieve the assembly's functional needs, the required tolerances will be tighter or wider, which will determine whether additional post-processes, such as machining, are required to achieve suitable accuracy.

Standard Fits

There are international standards in the metric system—ISO 286 and ANSI B4.2—and the imperial system—ANSI B4.1—that define the allowable tolerance limits that should be used depending on the type of fit required.

It is common to use the International Tolerance Grades defined in ISO 286, which provide a reference for typical manufacturing process capability in terms of tolerance accuracy, as shown in the following table:

Table 1: IT grades

Each IT grade establishes the allowable tolerance limits for a given dimension. As shown in the following table, a smaller IT grade provides tighter tolerances:

Standard tolerance grades	Values of standard tolerance (mm)								Nominal size (mm)
	from: 1 to: 3	3 6	6 10	10 18	18 30	30 50	50 80	80 120	
1	0,0015	0,0015	0,0015	0,0015	0,0015	0,002	0,002	0,003	Measuring tools
2	0,002	0,002	0,002	0,002	0,002	0,003	0,003	0,004	
3	0,003	0,003	0,003	0,003	0,004	0,004	0,005	0,006	
4	0,004	0,004	0,004	0,005	0,006	0,007	0,008	0,010	
5	0,005	0,005	0,006	0,008	0,009	0,011	0,013	0,015	Engineering fits, bearings, machining processes (grinding, turning)
6	0,007	0,008	0,009	0,011	0,013	0,016	0,019	0,022	
7	0,009	0,012	0,015	0,018	0,021	0,025	0,030	0,035	
8	0,014	0,018	0,022	0,027	0,033	0,039	0,046	0,054	
9	0,025	0,030	0,036	0,043	0,052	0,062	0,074	0,087	
10	0,040	0,048	0,058	0,070	0,084	0,100	0,120	0,140	
11	0,060	0,075	0,090	0,110	0,130	0,160	0,190	0,220	
12	0,090	0,120	0,150	0,180	0,210	0,250	0,300	0,350	Large manufacturing, die casting, stamping, sand casting
13	0,140	0,180	0,220	0,270	0,330	0,390	0,460	0,540	
14	0,250	0,300	0,360	0,430	0,520	0,620	0,740	0,870	
15	0,400	0,480	0,580	0,700	0,840	1,000	1,200	1,400	
16	0,600	0,750	0,900	1,100	1,300	1,600	1,900	2,200	
17	0,900	1,200	1,500	1,800	2,100	2,500	3,000	3,500	
18	1,400	1,800	2,200	2,700	3,300	3,900	4,600	5,400	

Table 2: Standard tolerance grades

Usually, the most common types of fits require very tight tolerances that cannot be achieved by designing and printing the part directly, and additional post-processes, such as machining, are required to achieve suitable accuracy.

Thus, there are some recommendations for designing a part that will need to be machined after printing to achieve the tight tolerances required. These recommendations include accurate holes and bearing housings.

Accurate Holes

Depending on how the hole is machined, a pre-hole or pilot hole can be designed into the part to guide the drill bit to the appropriated location. If the part is machined directly with a drill bit size equal to the final required hole diameter, it is recommended to machine the part without a designed pre-hole, letting the CNC Machine create the pre-hole to ensure proper positioning of the drill.

When it is necessary to machine a hole with a larger diameter than the available drill head, it will need to be machined by interpolating. In this case, a pre-hole or pilot hole can be designed into the part, where the required diameter must be at least 1 mm smaller than the final hole diameter.

Bearing Housings

In applications where fitting a bearing is required, it is recommended to machine it, interpolating with a smaller drill and then adjusting it to the required tolerance. Like the case mentioned above, a pre-hole can be designed to save material, where the required diameter must be at least 1 mm smaller than the final diameter to ensure a proper finish.

Figure 3: Bearings inserted into an HP MJF part

Customized Fits

When designing mating parts to create functional assemblies with a non-required standard fit, it is recommended to consider the following design guidelines:

• To print a clearance fit: When inserting a metal shaft into an HP MJF part hole, the minimum clearance must be as follows:

Clearance between mating parts >maximum size of the metal shaft + minimum size of the HP MJF part hole

• To print an interference fit: When inserting a metal pin into an HP MJF part hole, the minimum interference must be as follows:

Interference between mating parts >minimum size of metal pin+ maximum size of the HP MJF part hole

Designing MJF Threads

Introduction

The most widely used types of joints are screws and threaded parts because they can be disassembled several times and create strong and durable joints. The use of threads in plastic parts is common in the design of caps and customized fasteners or to join tubes.

General Recommendations

HP Multi Jet Fusion technology allows users to print external and internal threads inside the part, eliminating the need for mechanical thread-forming operations.

It is recommended to print external and internal threads in sizes larger than 6 mm (M6 or ¼ inch per the Imperial system) to achieve favorable results in all printing orientations. If a small thread (less than 6 mm) is needed, it is recommended to use self-tapping screws, threaded inserts, or to machine the thread for the small tolerances required in these sizes.

Jet Fusion Thread Guidelines

Self-Tapping Screws

Although HP Multi Jet Fusion technology allows for the printing of small features such as external and internal threads inside the part, when a small thread (up to 6 mm) is needed, it is recommended to use self-tapping screws, which tap their own threads as they are driven into the part. Certain types of self-tapping screws require a pre-formed hole, the dimensions of which can be recommended by the screw supplier.

Figure 1: Self-tapping screw

Machined Threads

Another alternative when a small thread (up to 6 mm) is needed is to machine the part after printing it in order to achieve the required accuracy. The tools recommended for machining HP Multi Jet Fusion parts are the same as other technical plastics.

Although not recommended, tools for machining metals like steel or aluminum may also be used.

Internal Threads

To machine an internal thread, it is necessary to start from a pre-formed hole and then machine the thread using the required tap. To design the pre-hole on the printed part, designers can refer to usual drill size recommendations for plastic and metal. For example, drill size recommendations for metric plastic threads are shown in the following table:

1. ISO Standard metric threads

ISO Metric thread size	Drill size (mm)
M3	2,5
M4	3,3
M5	4,2
M6	5
M8	6,8
M10	8,5
M12	10,2
M16	14
M20	17,5
M24	21
M30	26,5
M36	32
M42	37,5
M48	43
M50	47
M56	50,5

2. ISO Fine metric threads

ISO Metric thread size	Drill size (mm)
M3 x 0,35	2,65
M4 x 0,5	3,5
M5 x 0,5	4,5
M6 x 0,75	5,2
M8 x 1	7
M10 x 1,25	8,8
M12 x 1,25	10,8
M16 x 1,5	15,4
M20 x 1,5	18,5
M24 x 2	22
M30 x 2	28
M36 x 3	33

3. Whitworth threads

Thread size (inches)	Drill size (mm) BSW	Drill size (mm) BSP
1/16	1,2	--
3/32	1,8	--
1/8	2,6	8,9
5/32	3,1	--
3/16	3,6	--
7/32	4,4	--
1/4	5,1	11,9
5/16	6,5	--
3/8	7,9	15,4
1/2	10,5	19
5/8	13,5	--
3/4	16,5	24,7
7/8	19,3	28,4
1	22	30,8
1 1/8	24,8	35,5
1 1/4	27,8	39,4
1 3/8	30,5	42
1 1/2	33,5	45,4

Table 1: Drill size recommendations for plastic threads

Usually when machining metal parts, a set of three taps must be used to complete the process. With HP plastic materials, this can be reduced and only the final tap is needed due to the low hardness of HP material compared with the steel material of the tap.

Figure 2: HP MJF pre-formed hole testing

External Threads

To machine an external thread, it is necessary to start from a solid printed cylinder and then machine the thread using the required die. The diameter of the cylinder to be machined must be slightly smaller than the die's major diameter. Typical cylinder diameter recommendations for plastic and metal are applicable.

Figure 3: Metric die

Standard Printed Threads

To ensure a satisfactory assembly operation with HP Multi Jet Fusion technology, there are a few recommendations when designing threads larger than 6 mm under international standards (e.g., DIN 13-1, ISO 965-2, ANSI/ASME B1.1). These international standards usually specify tolerances relative to diameter and pitch of a thread.

When designing internal threads, the less restrictive tolerance values (maximum tolerance values) should be used, and when designing external threads, more restrictive tolerance values (minimum tolerance values) should be used. For example, when designing metric threads under the ISO 965-2 standard—threads with general-purpose tolerances (6H-6g) and normal engagement length—the recommended design values are shown in the following table:

		Internal threads - 6H					External threads - 6g					
		ØD	Pitch Diameter Ø		Minor Diameter Ø		Major Diameter Ø		Pitch Diameter Ø		Minor Diameter Ø	
Thread	Pitch	min.	min.	max.	min.	max.	max.	min.	max.	min.	max.	min.
M8	1.25	8	7.188	7.348	6.647	6.912	7.972	7.760	7.160	7.042	6.438	6.272
M10	1.5	10	9.026	9.206	8.376	8.676	9.968	9.732	8.994	8.862	8.128	7.938
M12	1.75	12	10.863	11.063	10.106	10.441	11.966	11.701	10.829	10.679	9.819	9.602
M16	2	16	14.701	14.913	13.835	14.210	15.962	15.682	14.663	14.503	13.508	13.271
M20	2.5	20	18.376	18.600	17.294	17.744	19.958	19.623	18.334	18.164	16.891	16.625

Table 2: Recommended external and internal thread tolerances for HP MJF, based on ISO 965-2 tolerances
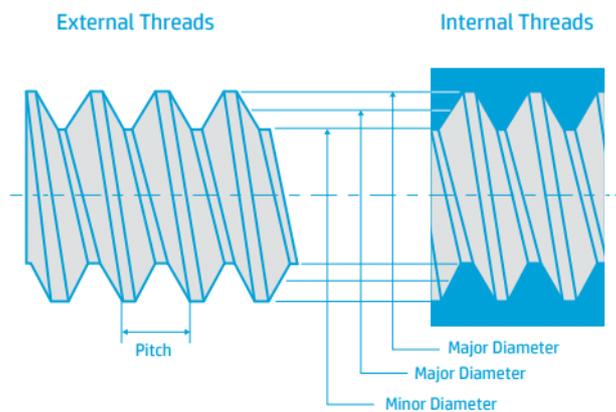
Figure 4: Features of external and internal threads

Customized Threads

For customized threads, all external and internal threads should be designed with a gap of 0.2 mm to 0.4 mm between the external and the internal thread, as appropriate.

It is recommended to remove all sharp edges and apply a minimum radius of 0.1 mm when designing threads for HP Multi Jet Fusion parts.
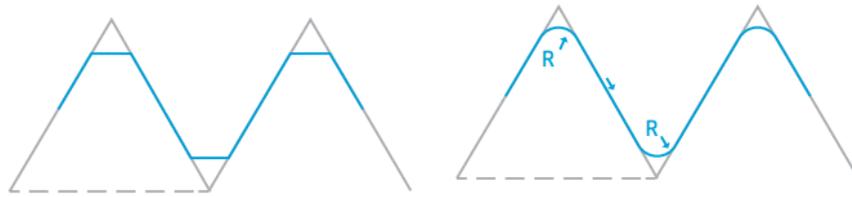
Figure 5: Round edges for custom-printed threads

Post-Processing Guidelines

Threads can be considered a very fine detail and should be cleaned using a manual or automatic sandblasting machine with glass bead particles that range from 70 to 110 microns in size and 3 to 4 bars in terms of pressure. For cases in which a vibratory finishing (tumbling) is required to improve surface roughness in other areas, it is recommended to first clean the threads using a sandblasting machine. Usually the media used in vibratory finishing are too big to clean the space between the threads.

It is more difficult to clean internal threads; for this reason, it is better to reduce the length of this type of thread and make through holes when possible. For internal and external threads, it is possible to use taps and dies if they are not completely cleaned or if there is excessive friction.

Painting the threads is not recommended in any case; parts can be painted only if they are already assembled.

For this reason, dying is the best option for coloring threaded parts without altering the dimensional accuracy.

Designing for Threaded Inserts in 3D Printed Nylon

Introduction

Threaded inserts provide a strong, reusable, and permanent thread in plastic parts, and they are typically used when frequent assembly and disassembly are required for service or repair. Threaded inserts are often available in brass, stainless steel, and aluminum, and can be installed using various techniques (e.g., heat-staking, ultrasonic vibrations, or press-in).

Recommended Threaded Inserts for HP Multi Jet Fusion

Selecting the best threaded insert type and installation technique depends on a few factors, such as part application, plastic part material, and strength requirements.

HP Multi Jet Fusion parts are made of thermoplastic materials and can be re-melted and re-formed once printed. For this reason, inserts that are installed by heat-staking and ultrasonic vibrations are recommended for thermoplastic materials due to their high overall performance; however, press-in (screw-to-expand or hexagonal-shaped) and self-threading inserts may also be used in some applications.

Type of insert		Performance
Heat-staking and ultrasonic vibrations insert		High overall performance. Not very dependent on hole size. Material is melted around the insert.
Press-in insert	Screw-to-expand	Very dependent on hole size tolerances. Recommended for non-critical applications.
	Hexagonal-shaped	Dependent on hole size tolerances. Good pull-out resistance. Recommended for non-critical applications.
Self-threading insert		Excellence pull-out resistance. Easy to install.

Table 1: Types of inserts

Guidelines for Using Threaded Inserts in MJF Nylon

Hole Diameter

A pre-formed hole is necessary to install a threaded insert, so the hole diameter is a very important element in achieving the desired strength: Oversized holes will result in a reduction of the joint strength and undersized holes can potentially crack the part. Usually, suppliers of threaded inserts specify the hole diameter size and depth needed to install an insert.

> The hole size is dependent upon the part orientation, and for this reason it is recommended to always print the part using the same orientation.

HP Multi Jet Fusion–produced parts may have dimensional variations in small features up to +/- 0.2 mm (ISO 286, IT Grade 13), which is usually higher than supplier specifications. For this reason, it is important to select a type of insert that is compliant with hole deviations.
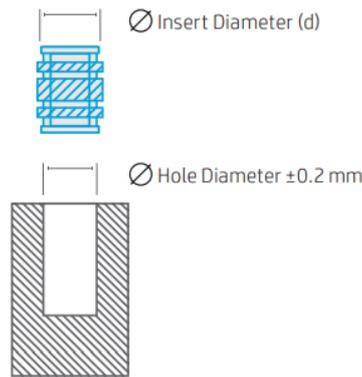
Figure 1: Hole diameter

Bosses

Bosses are typically used for mounting purposes such as attaching fasteners or as a receptacle for threaded inserts. Traditionally a boss diameter is twice the size of the external diameter of inserts that are less than 6 mm, while a 3-mm wall thickness applies to all larger inserts.
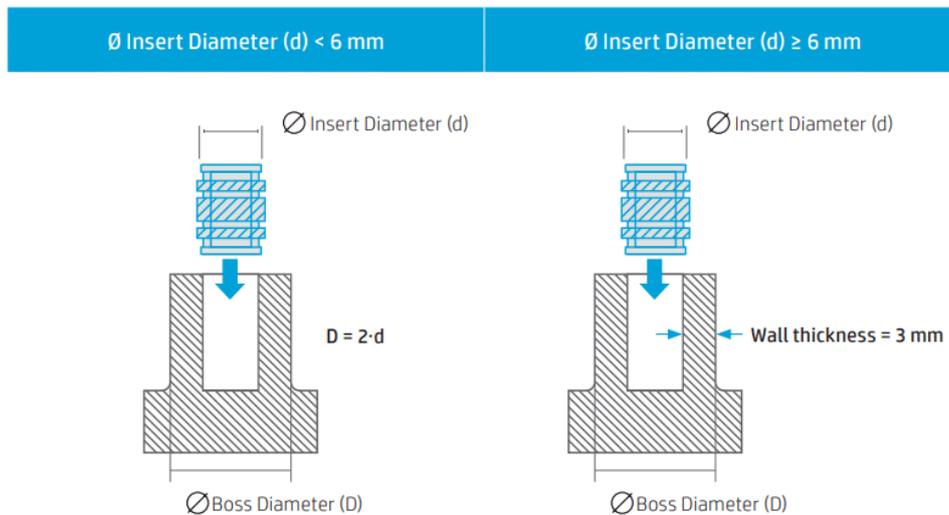
Figure 2: Boss diameter

Special consideration should be given to cold-press installations where increased stress may require larger boss diameters.

Mating Parts

The threaded insert—not the plastic part—should bear the load. For this reason, the diameter of the mating part hole is also important to keep the insert from being pulled through the hole.

Thus, the diameter of the mating part hole must be larger than the outside diameter of the assembly bolt but smaller than the diameter of the insert, as shown below:
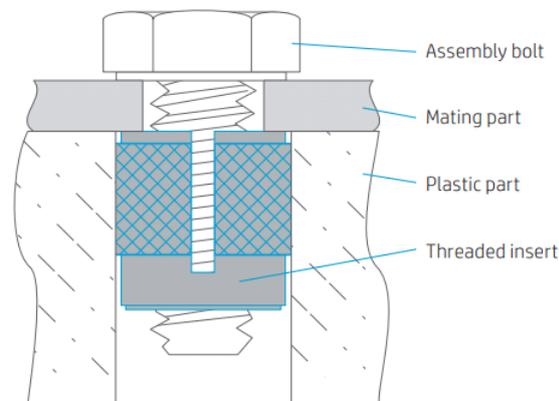
Figure 3: Threaded joint assembly

The mating part must also withstand the stress generated by the clamping force. In instances where the mating part will also be plastic, the use of a collar or a washer between the assembly bolt and the mating part should be considered.

Design to Split and Adhesive Bond 3D Prints

Introduction

When using HP Multi Jet Fusion technology, it is sometimes necessary to split a part into different pieces and then re-join them.

There are two main reasons why bonding parts together may be necessary:

Some big parts do not fit inside the build chamber of HP Jet Fusion 3D printers. Therefore, the parts can be split into several pieces and then re-assembled after printing. This can occur in the automotive industry or in applications such as jigs and fixtures, where big parts could require bonding to ensure a strong joint and achieve a proper solution.

Splitting Big Parts
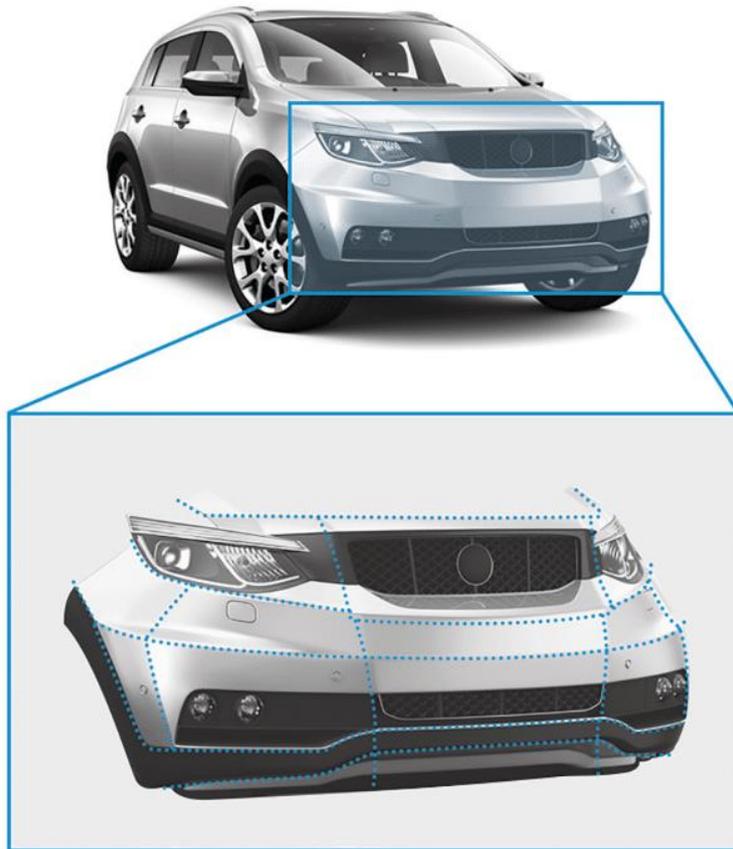
Figure 1: Bumper split proposal

Increasing Packing Density

The maximum printing efficiency in terms of cost and productivity is achieved by increasing the packing density. Depending on the geometry, there may be packing limitations for the achievable maximum value. In these cases, splitting the parts is a possible option.

For example, a part's packing density could be optimized by adding hinges that allow it to fold. These could be blocked after printing by using an adhesive or other mechanical locker.
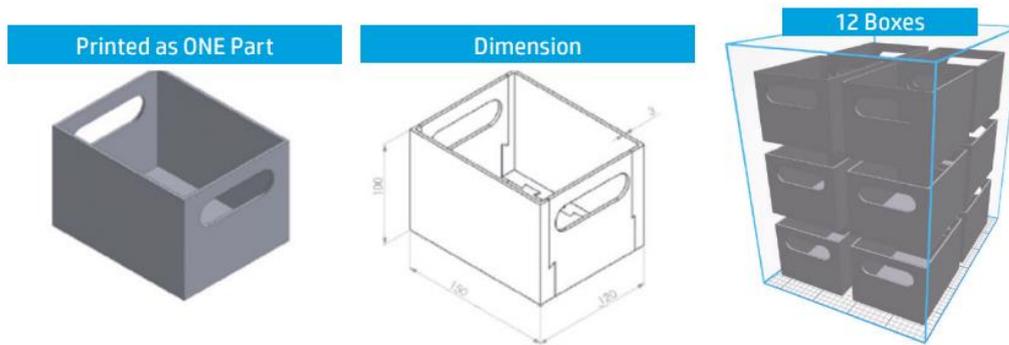
Figure 2: Example of packing density optimization: Box original design and number of parts that fit in the print bed. Data courtesy of Henkel AG & Co. KgaA
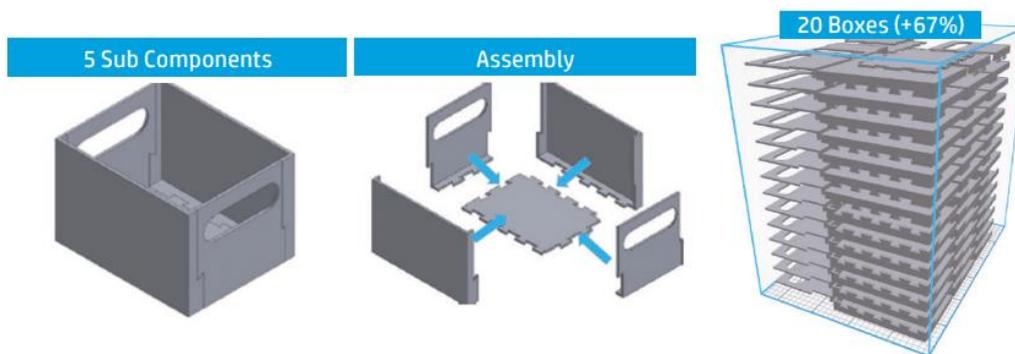
Figure 3: Example of split part to optimize packing density: With the split, the packing density increased by 67%. Data courtesy of Henkel AG & Co. KgaA

Splitting and Adhesive Bonding MJF Guidelines

Bonding robustness depends highly on the design of the union and the way the part has been split or cut into different pieces. A proper bonding design is critical for success.

Union Design

The union design of the bonding is key to ensure proper performance of the bonding in the final part. The time invested in designing a proper union may depend on the final use of the pieces that will be bonded. For example, a visual prototype that will not withstand any loads would require a simple design union, while an automotive part that will be included in the final product should be designed to optimize its performance.

Design options depend on the thickness of the bonded parts and on the possibility of modifying the final geometry.

Thickness < 1.7 mm with no geometry modification allowed.

One of the objectives in the design of the union is to increase the bond area as much as possible. Including features that will help reference one piece to the other during bonding will help achieve the proper position between the parts and will optimize the final result. The most recommended design option for this case is a dove or jigsaw feature, as shown below:

Dove/Jigsaw
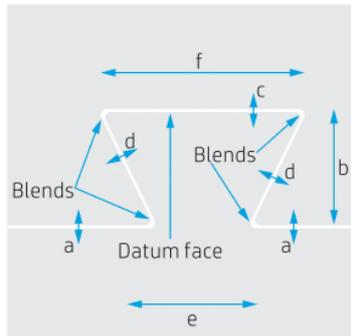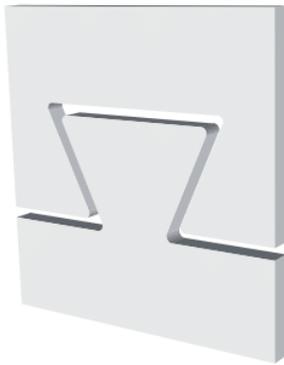
Recommended dimensions:

a: 0.2 mm
b: 10 mm
c: 0–0.1 mm
d: 0.2 mm
e: 10 mm
f: 20 mm

Figure 4: Dove/jigsaw design recommendations

This type of union will help increase the bonding area and, at the same time, it positions and holds both pieces that will be assembled.

There are also other design options that are simpler and will also provide satisfactory results, which could be an option for faster designs:

Square tongue
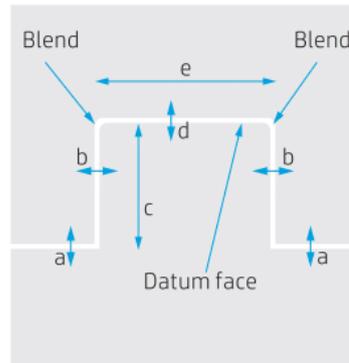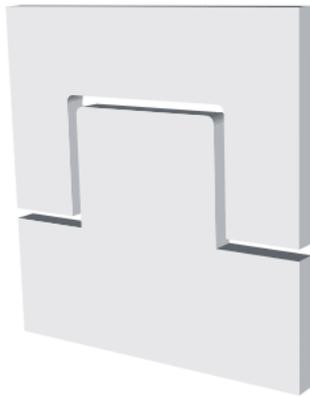
Recommended dimensions:

a:	0.4 mm
b:	0.2 mm
c:	10 mm
d:	0-0.1 mm
e:	10 mm

Figure 5: Square tongue design recommendations

Tooth
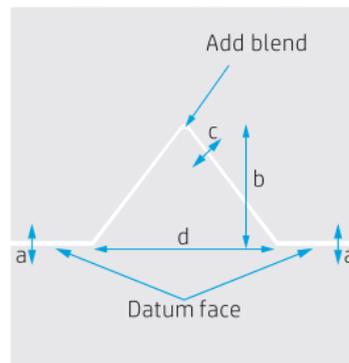
Recommended dimensions:

a:	0-0.1 mm
b:	10 mm
c:	0.2 mm
d:	20 mm

Figure 6: Tooth design recommendations

Butt
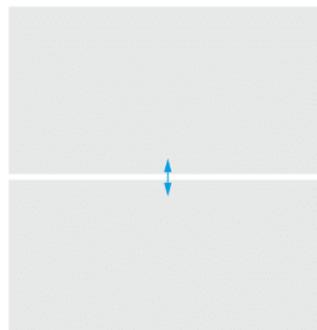
Recommended gap between parts:

0-0.1 mm

Figure 7: Butt design recommendations

When adding a union design that is not viable due to geometrical constraints (such as a dove, square tongues, or a tooth), a butt union design could be an alternative, bearing in mind that having a straight line in the bonding area is the weakest union design option due to its less-available bonding area.

Thickness < 1.7 mm with geometry modification allowed

If a modification in the geometry is allowed, the bonding area can be increased and mechanically reinforced by adding an
overlap between the bonded areas.
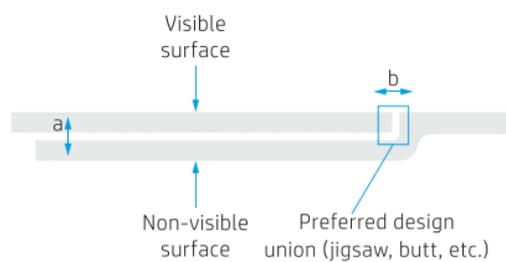
Figure 8: Offset overlap design recommendations

Thickness > 1.7 mm

When there is adequate thickness to add an overlap between the parts, it is no longer necessary to modify the final geometry to reinforce it. The improvement and optimization of the bonding union can be executed directly in the original design, thus increasing and reinforcing the bonding area.
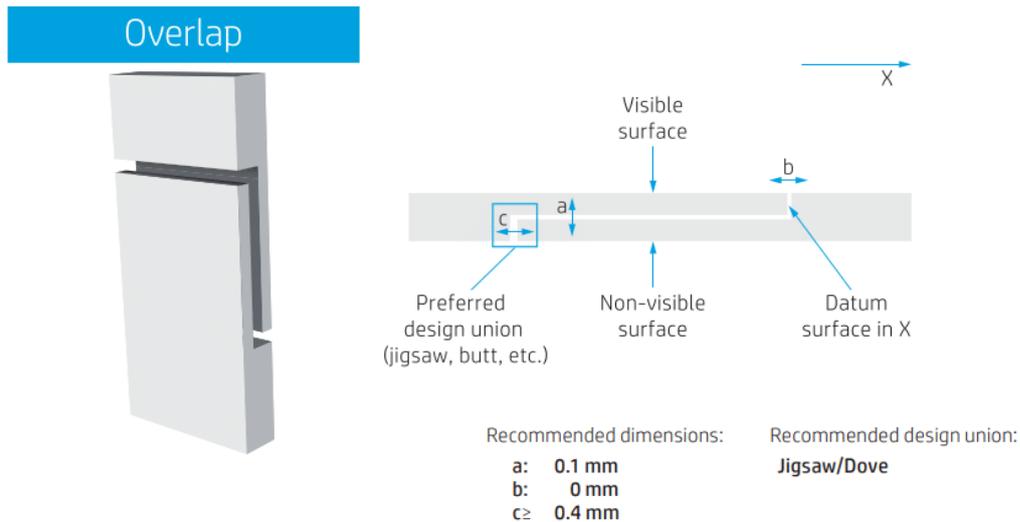
Figure 9: Overlap design recommendations

Adding more than one union feature

When the bonding line is long, it may be helpful to add multiple features that will hold both pieces together when the adhesive is applied. Regardless of whether the position between both parts is critical to the design, the features design must be executed by first adding a reference feature that will position both parts in the XY plane, and then by adding the remaining features with a higher clearance in order to absorb any dimensional variation.
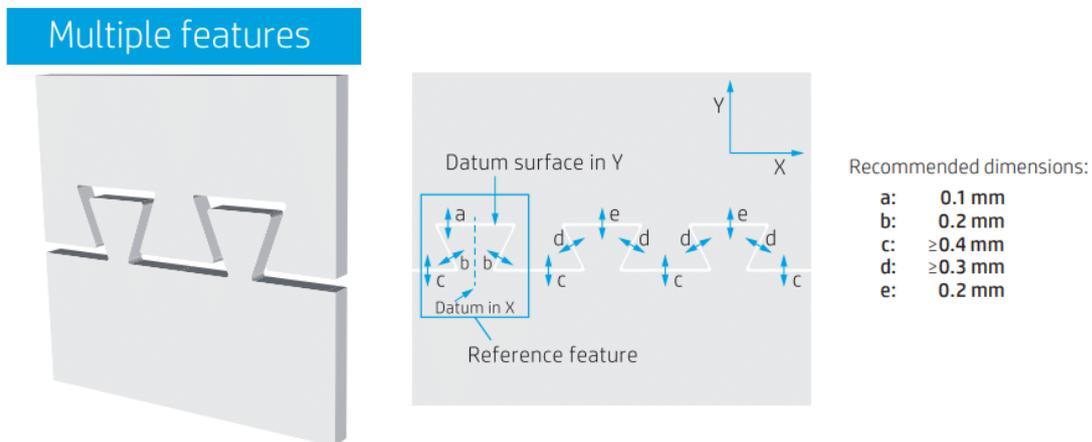
Figure 10: Multiple union features: Design recommendations

Combination of overlap joint with multiple jigsaw features

When taking into account all of the design recommendations mentioned above, the preferred union design is the combination of the overlap joint with reference features such as the jigsaw. Those reference features can be added to the non-visible surface and will help reference the two pieces between them to optimize the bonding performance.

Additional Options to Increase Bonding Adhesion

When the adhesive material has the ability to fill gaps, the mechanical adhesion between the adhesive and the bonding parts can be improved by adding textures to their surfaces. This roughness improvement allows for mechanical interlocking by adding "teeth" to the surface and increases the total effective bonding area.

Figure 12: Example of texture to increase adhesion

An additional option to increase the bonding area is to include grooves to the bonding parts' surfaces.

Recommended dimensions:

a: 0.2 mm
b: 0.1 mm
c: ≥1 mm
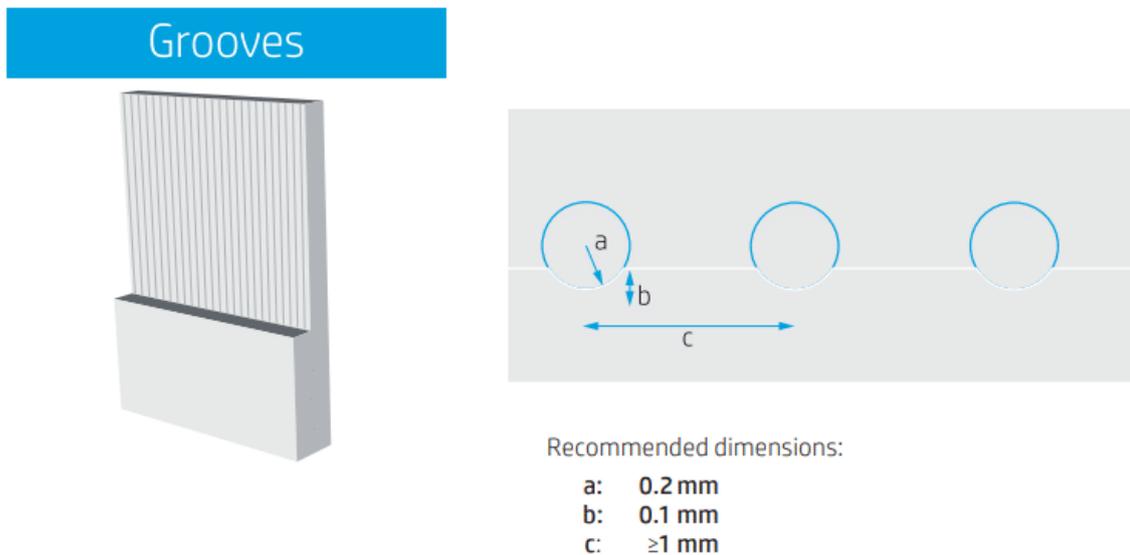
Figure 13: Grooves to increase bonding area: Design recommendations

Things to Consider when Splitting your Part during File Preparation

When a part needs to be cut, bear in mind which parts are the loads that will be applied to the final bonded assembly, as the bond robustness can be highly optimized depending on the design decisions. The stress that appears in the bonding area depends on the applied loads. The most common are as follows:
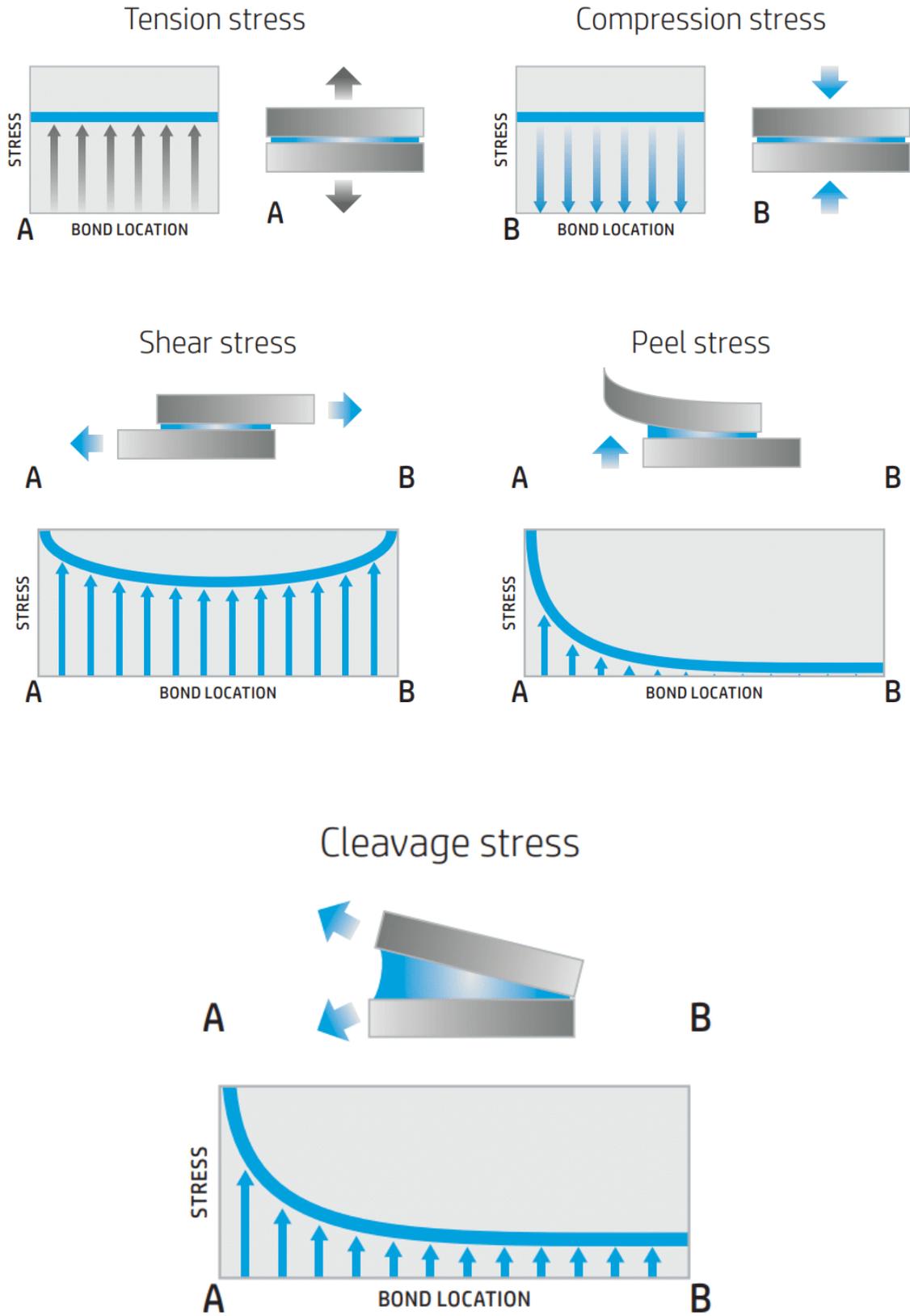
Figure 14: Stress distribution generated by different loads. Data courtesy of Henkel AG & Co. KgaA

When the cut is created, the design should be finished in order to prevent the development of peel, cleavage, or tension stress. The adhesive bondings work better under shear or compression stress, and the design should maximize the generation of those types of stresses to achieve a robust joint. The introduction of the overlap in the joint helps develop a bonding that works better under shear stress, which maximizes the performance of the union.

It is recommended to have a wider overlap area rather than a longer one. Having a wider overlap helps distribute more stress along the width and reduces the maximum values that appear on the sides.

Designing 3D Printed Snap Fits

Introduction

A snap-fit is an efficient assembly method used to attach plastic parts via a protruding feature on one part (e.g., a hook), which
deflects during assembly to be inserted into a groove or a slot in the second part. After the assembly, the protruding feature
returns to its initial position.
Snap-fits provide a simple and economical way to assemble plastic parts by drastically reducing assembly time. The way a
snap-fit is designed determines whether it can be disassembled and reassembled several times and the force required to do
so. This assembly method is suited to thermoplastic materials for their flexibility, high elongation, and ability to be printed into
complex shapes.
HP Multi Jet Fusion technology allows for the designing and printing of parts with specific design features integrated, such as
snap-fits, in order to connect them.

Types of Snap-Fits

The various types of snap-fits are listed below.

Cantilever Snap-Fit

The cantilever snap-fit is the most commonly used type of snap-fit. It consists of a cantilever beam with an overhang at the end. In this type of snap-fit there is a direct relationship between the robustness of the assembly and the strength of the snap-fit.
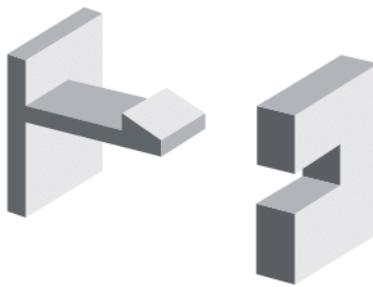
Figure 1. Cantilever snap-fit

Snap not actuated

Snap being actuated

Snap actuated

Figure 2. Cantilever snap-fit assembly operation

L-Shaped Snap-Fit

When it is not possible to design a cantilever snap-fit without compromising the robustness of the assembly and the strength of the snap-fit due to material or geometrical constraints, an L-shaped snap-fit can be an alternative. Adding a groove to the base of the snap-fit increases its flexibility while reducing the strain on the beam, compared with a cantilever snap-fit.
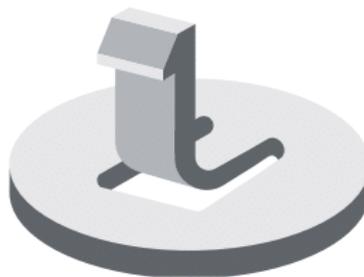
Figure 3. L-shaped snap-fit
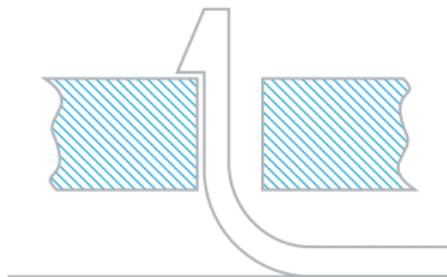
Figure 4. L-shaped snap-fit assembly operation

U-Shaped Snap-Fit

The U-shaped snap-fit is another alternative to the cantilever snap-fit when it is necessary to increase the snap-fit flexibility
within a reduced space. This U-shaped alternative is extremely flexible, and thus easier to remove. This type of snap-fit is
usually used in cases where the parts need to be pulled apart repeatedly or when two parts don't require a lot of force to stay
in position (e.g., in a battery compartment lid).
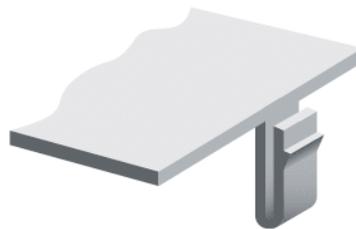
Figure 5. U-shaped snap-fit
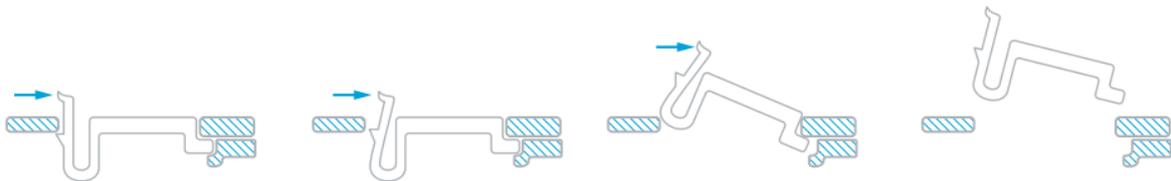
Figure 6. U-shaped snap-fit assembly operation

Annular Snap-Fit

The annular snap-fit is an assembly method usually used between two cylindrical or ring-shaped parts or between two rotationally symmetric parts, where the deformation required to assemble or disassemble the snap-fit is made in a 360º direction at the same time.

With this assembly method, one part is designed with an undercut and the other is designed with a mating lip. The joint occurs through the interference between both parts during the assembly operation.
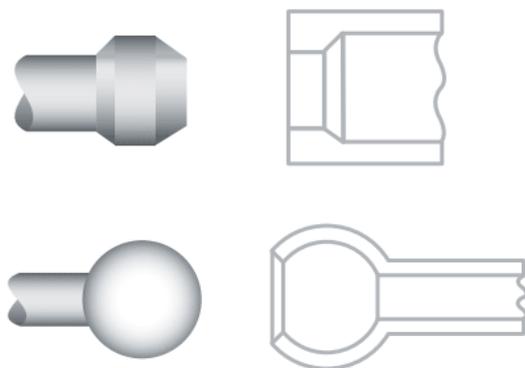
Figure 7. Annular snap-fit
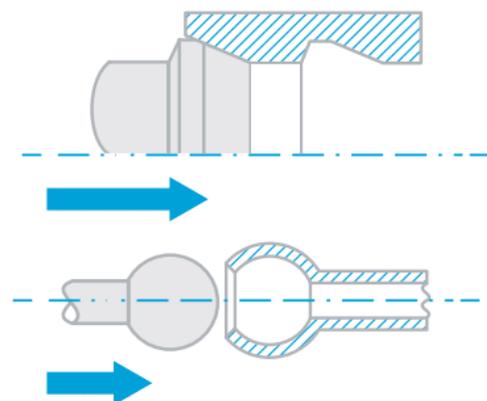
Figure 8. Annular snap-fit assembly operation

Torsional Snap-Fit

The torsional snap-fit is an assembly method where the flexible point is in a torsional bar instead of the self–snap-fit body.

When the torsional bar is pushed down, it turns slightly and opens the joint.
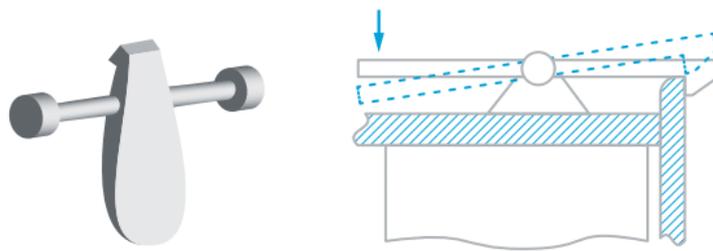
Figure 9. Torsional snap-fit

Snap Fit Design Considerations

As mentioned previously, the most commonly used type of snap-fit is the cantilever snap-fit. When designing this type of snap-fit, it is important to design a balanced solution between the robustness of the assembly and the strength of the snap-fit cantilever beam.

This type of snap-fit can be approximated using a simplification of the general beam bending theory, which allows for the inspection of the snap-fit design feasibility. This approach models the cantilever snap-fit by a fixed-free beam with a point applied end load:

Figure 10. Cantilever beam with a point-applied end load

Mating Force and Beam Stress

The robustness of the assembly will be defined by the force (P) required to assemble and disassemble it. A weak force required to deflect the snap-fit beam will lead to a weak assembly that is unable to maintain the connection between both parts. Otherwise, a strong force will lead to an extremely robust assembly, which will be difficult to assemble and disassemble when required.

Moreover, the design of the snap-fit must be strong enough to resist the stress (σ) suffered by the beam when it deflects due to the mating force (P) applied, without compromising the snap-fit integrity and performance.

For this reason, the mating force (P) and the beam stress (σ) must be the main considerations when designing a cantilever snap fit, and according to the beam bending theory, they are dependent upon the snap-fit geometry and the material used to make it.

Material and Geometry Dependence

Because of their direct relationship with the assembly robustness and snap-fit strength, the snap-fit material and geometry are considered the most critical design parameters, and they are often dependent upon the available design space.

For this reason, geometry and material choice are usually the first steps when designing a snap-fit.
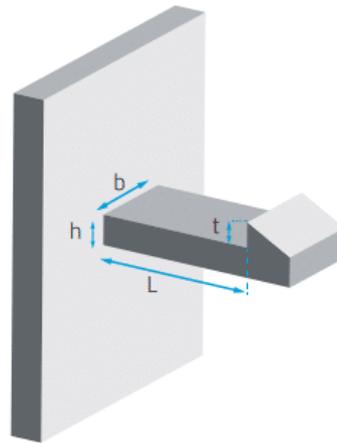
Figure 11. Snap-fit geometry

When choosing the snap-fit material and geometry (h, b, L, t), other dependent factors are clearly defined:

• Choosing the snap-fit cross-section geometry (h, b) allows the designer to calculate its moment of inertia (I), which, for a cantilever beam with a rectangular cross-section, is as follows:

$$I = \frac{b \cdot h^3}{12}$$

• Once the printing material is selected, the modulus of elasticity (E) is made clear since it is often provided in the materialdatasheet.

> The product of the moment of inertia (I) and the modulus of elasticity (E) is known as the beam flexural rigidity (EI).

According to the beam bending theory, these dependent parameters, along with the snap-fit material and geometry, have a direct relationship with the required mating force (P) and the beam stress (σ), as shown below:

• Deflection (y) at the end of a cantilever beam with a point-applied end load:

$$y = \frac{P \cdot L^3}{3 \cdot E \cdot I} \quad (1)$$

• Maximum stress (σ) in a cantilever beam with a uniform rectangular cross-section:

$$\sigma = \frac{P \cdot L \cdot h}{2 \cdot I} \quad (2)$$

The minimum amount of deflection (y) at the end of the cantilever beam required to assemble and disassemble the snap-fit is usually a known parameter dependent upon the geometric constraints and the available design space. In fact, it is defined by the depth (t) of the snap-fit overhang:

• The minimum amount of deflection (y) must be at least equal to the depth (t) of the snap-fit overhang to allow a proper assembly and disassembly operation.

$$y \geq t$$

• A deeper overhang will lead to a strong assembly, but it will mean that the beam must deflect further and, as a consequence, it will require a greater matting force (P)—as shown in equation (1) —and the beam stress (σ) will also increase—as shown in equation (2)-.

Snap Fit Design Calculations

The first step in checking the snap-fit design feasibility is to calculate the resultant mating force (P) and to check whether it is suitable. This calculation can be done by solving the equation (1) for P:

$$P = \frac{3 \cdot E \cdot I \cdot y}{L^3} \quad (3)$$

Based on the equation (3), the force (P) is dependent upon how much farther the snap-fit beam must deflect (y), but it also will depend on the material resistance against the bending deformation, which is known as beam bending stiffness (k), and its unction of the beam flexural rigidity (EI), the length (L) of the beam, and beam boundary condition:

$$P = k \cdot y \quad (4)$$

The suitable mating force (P) value should not be greater than 50N to 100N, which is considered an ergonomic value for an estimated finger strength average.

Once the mating force (P) has been calculated and it results in a suitable value, the second step to check the snap-fit feasibility is to calculate the stress (σ) in the cantilever beam based on the equation (2).

If the beam stress (σ) is above the yield strength of the material, the snap-fit will deform, and some part of the deformation will be permanent and non-reversible, thus compromising the snap-fit performance and strength up to rupture.

Beam stress (σ) < Material yield strength (5)

Considering that the yield strength is not a common property specified in technical datasheets when producing plastic parts, the best option to calculate the snap-fit strength is to use the material allowable strain (ε) and modulus of elasticity (E):

Beam stress (σ) < E · ε (6)

In order to obtain the allowable strain (ε) value, designers can refer to usual recommendations for other plastic manufacturing processes such as Injection Molding:

Allowable strain (ε) < 1/3 · Material elongation at yield

All design considerations are shown in the following flowchart:
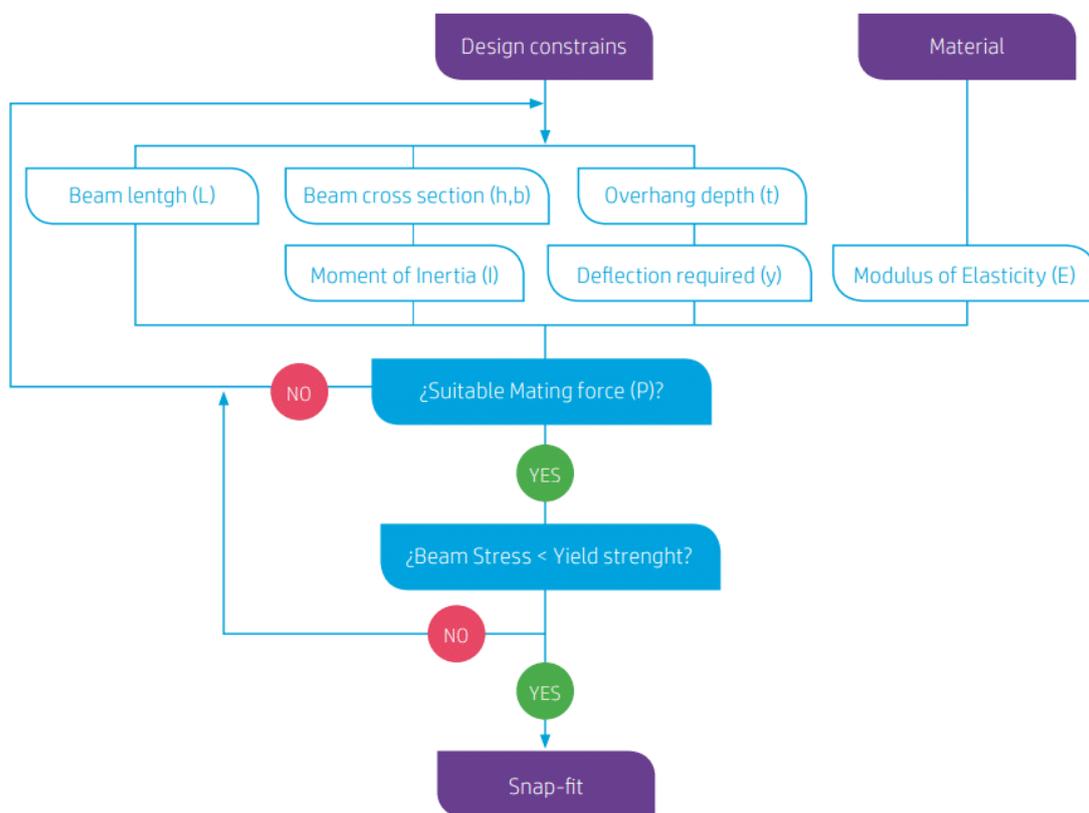
Figure 12. Design snap-fit flowchart

Figure 12. Design snap-fit flowchart

	(Permissible) deflection			Deflection force
Type of design				
Shape of the cross section				
 Rectangle	$y = 0.67 \cdot \dfrac{\varepsilon \cdot L^2}{h}$	$y = 1.09 \cdot \dfrac{\varepsilon \cdot L^2}{h}$	$y = 0.86 \cdot \dfrac{\varepsilon \cdot L^2}{h}$	$P = \dfrac{bh^2}{6} \cdot \dfrac{E\,\varepsilon}{L}$
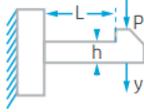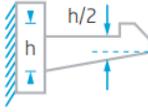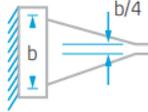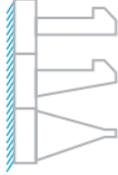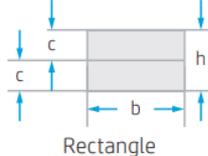
Figure 13. Snap-fit calculation equations*

*Note 1: Beam deflection (y) expressed in terms of allowable strain (ε), based on equations (1), (2)
*Note 2: Mating or deflection force (P) expressed in terms of allowable strain (ε), based on equation (2)

Design guidelines

There are several design recommendations when designing snap-fits with HP Multi Jet Fusion:

Minimum Thickness (h)

The minimum recommended thickness at the base of the cantilever is 1 mm.
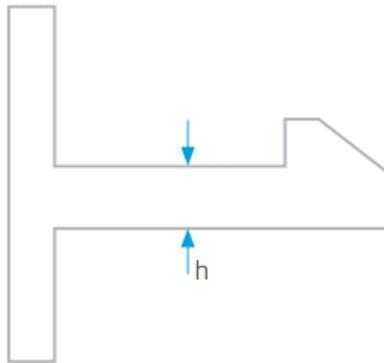
Figure 14. Minimum thickness at the base of the cantilever

Minimum Overhang Depth (t)

The minimum overhang depth (t) should be at least 1 mm.
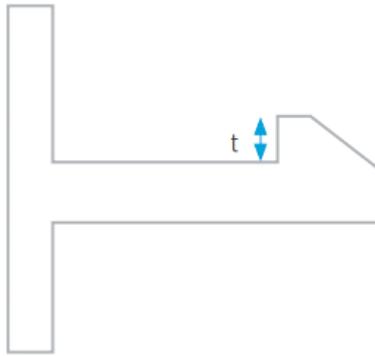
Figure 15. Minimum overhang depth (t)

Recommended Common Radius

It is recommended to add a common radius at the base of the cantilever to avoid sharp corners and reduce the stress concentration. This common radius should be at least half of the thickness (h) of the base of the cantilever.
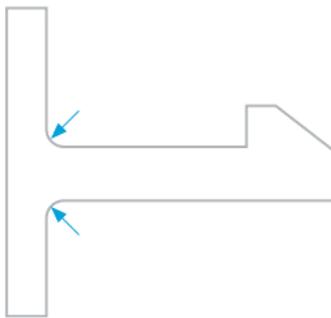
Figure 16. Common radius

Snap-Fit Overhang

It is recommended to avoid sharp edges at the end of the snap-fit overhang, adding a small chamfer to prevent breaking during the assembly operation.
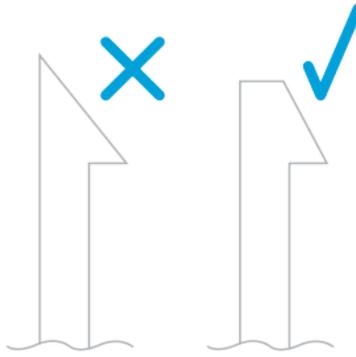
Figure 17. Snap-fit overhang

Assembly Angle (α)

As mentioned previously, the snap-fit overhang usually has a gentle chamfer to facilitate the assembly operation. The inclination of this chamfer angle (α) directly affects the mating force (P). If the angle (α) is reduced, the mating force (P) will also reduce. The recommended assembly angle value should be between 35º and 40º.
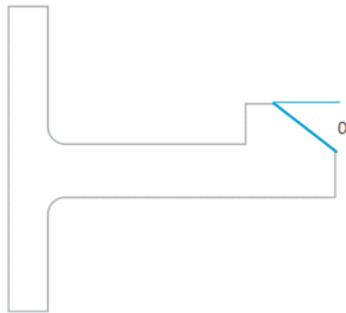
Figure 18. Assembly angle

Disassembly Angle (β)

The way the overhang is designed determines whether the snap-fit can be disassembled and reassembled several times. The disassembly angle (β) affects the ease of joint disassembly. For example, a 90º angle (β) can never be disassembled. However,
a snap-fit with a disassembly angle (β) equal to the assembly angle (α) will need the same mating force (P) for both operations.
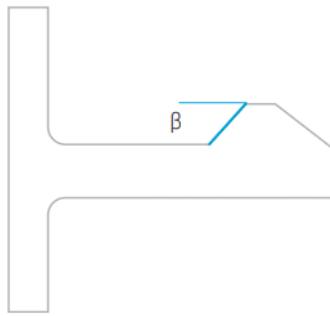
Figure 19. Disassembly angle

Tolerances between Parts

When designing a snap-fit, there must be a gap between the protruding feature and the groove to ensure a proper performance, even including the worst tolerance case as shown in the following figure:

$L_A \pm d_A$ $L_B \pm d_B$

$$L_A \geq L_B + d_B + d_A$$
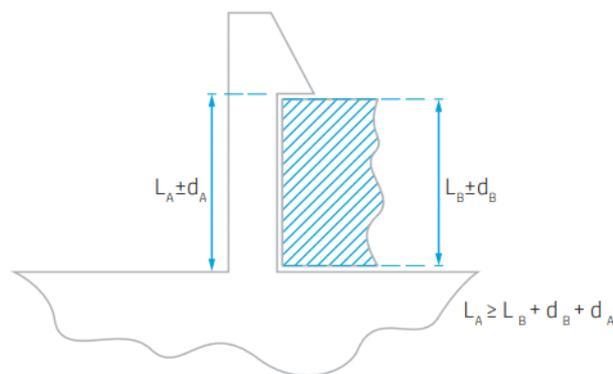
Figure 20. Tolerances between parts

Modifying the Mating Force (P)

Sometimes, after choosing the snap-fit material and geometry, the resulting mating force (P) is a non-desirable value. Based on the equation (3) and bearing in mind that when designing a snap-fit, the most common restrictive tolerances are the length (L) of the beam and the depth (t) of the overhang, the most common solution when modifying the mating force (P) is needed is to change the cantilever cross-section (h, b).

Reducing the mating force (P) will also reduce the beam stress (σ).

Tapered Beam

One of the most recommended changes in the snap-fit cross-section is to design a tapered beam. While a snap-fit beam with a uniform cross-section has an uneven distribution of strain and concentrates the stress at its base, a tapered beam uses less material and results in a more even distribution of strain throughout the cantilever, thus reducing stress (σ) concentration and the assembly and disassembly force (P).
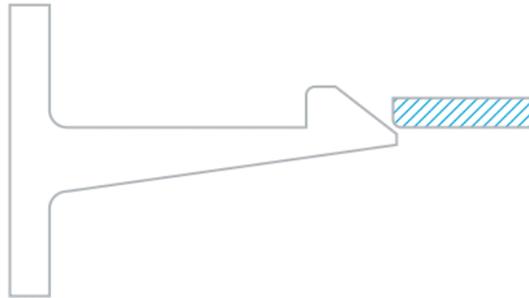

Figure 21. Tapered beam

Printing Orientation

There are some recommended orientations when printing a snap-fit regarding its accuracy and proper performance.

For Tight Snap-Fits

When printing tight snap-fits where the length of the beam (L) is critical, the XY plane orientation is recommended to achieve the best accuracy and, thus, a better performance.
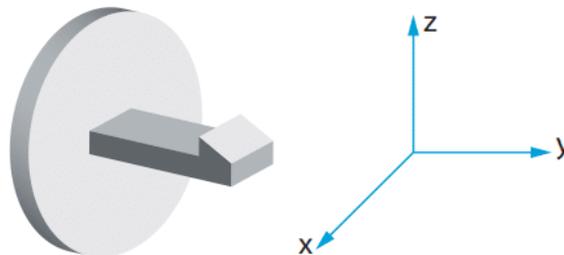
Figure 22. XY plane orientation

When the width of the snap-fit (b) is critical, the XZ or YZ plane orientation is recommended to achieve the best accuracy and to avoid excessive clearances on the XY plane, which can lead to noise and vibrations.
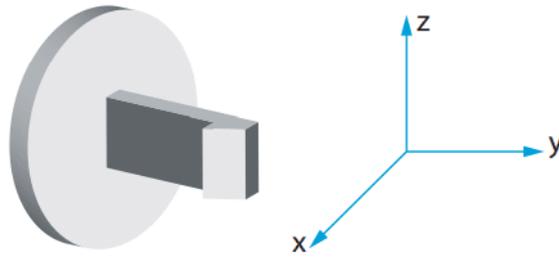
Figure 23. XZ or YZ plane orientation

To Reduce Printing Issues

Printing the snap-fit inclined slightly in the X, Y, and Z axes can reduce the likelihood of typical printing issues.

Post-Processing Recommendations

HP MJF technology allows for different post-processing methods that can affect the finishing of the printed part. Although most of the post-processing methods should not affect a 3D printed snap-fit, there can be some automatic post-processes that affect it, such as the tumbler post-process.

The tumbler post-process involves hitting the 3D printed part with small abrasive pellets in order to reduce its roughness. In return, some dimensions and/or small features can be affected by the process.

In the case of the snap-fits, a tumbler process can reduce the mating force (P) of the assembly and even break it depending on the snap-fit geometry.

For this reason, if automatic post-processes are required, it is recommended to protect the part with a sinter box to prevent damage.

Calculation example

The following figure illustrates the calculation needed when designing a cantilever snap-fit.

In this particular case, a clipping system for an optical sensor must be designed as follows:
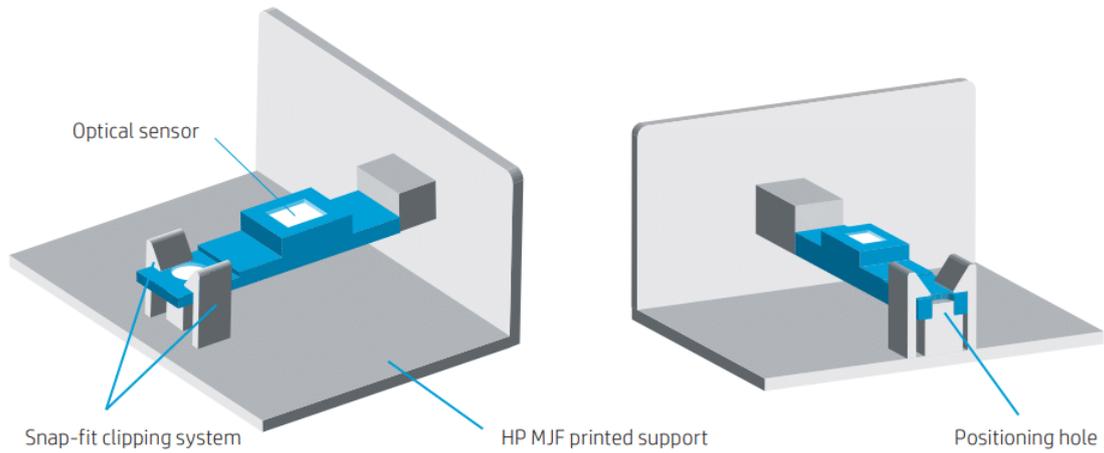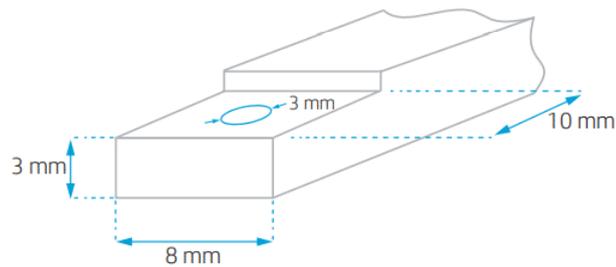
Optical sensor

Snap-fit clipping system

HP MJF printed support

Positioning hole

Figure 24. Optical sensor clipping system

3 mm

10 mm

3 mm

8 mm

The design requirements are listed below:

• The material used to print the part is HP 3D HR PA 12, with an elastic modulus of elasticity (E) of 1800 MPa.

• Due to optical requirements, the sensor must lay 5 mm above the base. Thus, the snap-fit total length must consider the worst-case tolerances and the optical requirements:

$$L = 3\ mm + 0.1\ mm + 5\ mm + 0.2\ mm + 0.2\ mm = 8.5\ mm$$

$L \pm 0.2\ mm$

3 mm ± 0.1 mm

5 mm ± 0.2 mm
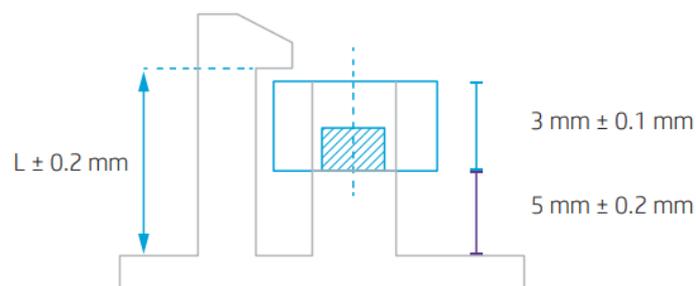
Figure 26. Snap-fit length calculation

• Due to constructive constraints, the snap-fit cannot overlap the positioning hole, which means that the overhang depth (t) must be between 1 mm—the minimum recommended value—and 2.5 mm—the maximum allowable distance to avoid contact between the snap-fit overhang and the sensor positioning hole:

Overhang depth (t) = y = 1 mm

• The width must be smaller than 10 mm due to geometrical constraints:

b = 9.5 mm

Once the snap-fit material and geometry (h, b, L, t) are clearly defined, the resulting mating force (P) must be calculated to check whether it is suitable. This calculation can be done using the equation (3):

$$P = \frac{3 \cdot E \cdot I \cdot y}{L^3} = \frac{3 \cdot E \cdot (b \cdot h^3) \cdot y}{12 \cdot L^3} = \frac{3 \cdot 1800 MPa \cdot 9.5\ mm \cdot (1.5\ mm)^3 \cdot 1\ mm}{12 \cdot (8.5\ mm)^3} = 23.49N$$

The calculated mating force (P) value is inside the ergonomic range. Therefore, based on the equation (2) and (6), the next step is to check the strength of the snap-fit calculating the allowable strain (ε):

$$\sigma = \frac{P \cdot L \cdot h}{2 \cdot I} = E \cdot \varepsilon$$

$$\varepsilon = \frac{P \cdot L \cdot h}{2 \cdot I \cdot E} = \frac{12 \cdot P \cdot L \cdot h}{2 \cdot (b \cdot h^3) \cdot E} = \frac{12 \cdot P \cdot L}{2 \cdot (b \cdot h^2) \cdot E} = \frac{12 \cdot 23.49N \cdot 8.5\ mm}{2 \cdot (9.5\ mm \cdot (1.5\ mm)^2) \cdot 1800\ MPa} = 0.03 = 3\%$$

The calculated allowable strain (ε) shows that the snap-fit does not deform when it deflects due to the mating force (P) applied,
without compromising its integrity and performance.

Recommended 3D CAD File Formats and Resolutions

Introduction

Before sending a job to print, the 3D model must be converted into a file extension that HP Jet Fusion 3D Printing Solutions are able to interpret.

The most commonly used file extension in 3D printing is STL, despite the fact that it lacks even basic 3D model information such as color or the identification of distance units.

For this reason, HP and other 3D printing leaders have identified the need to develop a file extension that supports the needs of current 3D printing applications, services, and devices. A 3MF consortium has been formed to deliver a format—3MF—that meets the needs of the 3D printing industry with the possibility to grow as the industry evolves.

Moreover, a 3MF extension file will have a much lower weight compared with an STL file for a specific 3D model and resolution.

Tessellation or Facets

To convert a 3D model into a 3D printing file, it is necessary to tessellate (or facet) the model, which means converting its geometry into linked triangles to convey its surface.

Once the 3D model has been tessellated, it is imported into slicer software, which slices the 3D model into layers and prepares it to be sent to the printer. It is very important to pay attention to this step: If it is not done correctly, it can cause problems such as geometric inaccuracies or slow processing.

A normal file size for a 3D model is between 1 and 30 MB, but the size depends on the type of software that created it, the number of triangles, and the amount and level of details (e.g., a higher resolution means in a higher number of triangles, which will result in a heavier file size).
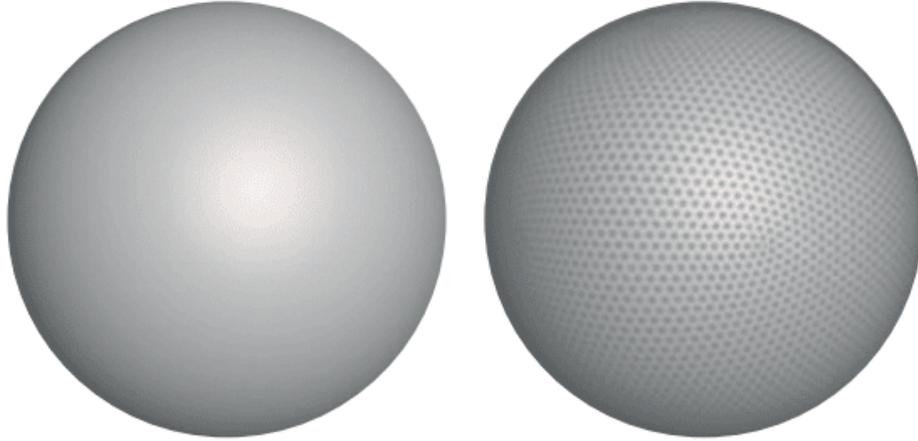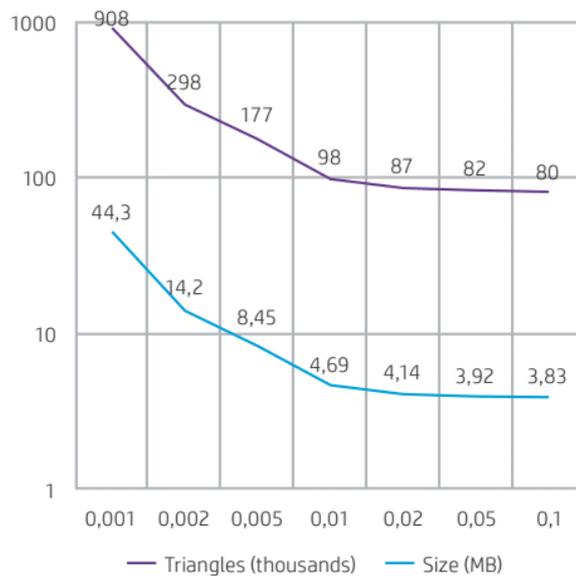

Figure 1. Example of tessellation

Figure 2. Triangle counts and file sizes vs. exported accuracy

Exporting a Model from CAD

Although each version of CAD software uses a different method to export a 3D model to an STL or 3MF file, it is often necessary to manually enter some exporting parameters such as deviation

chord height and angle tolerance, which define the resolution and the size of the STL or 3MF file by altering the tolerance in CAD software.

Deviation Chord Height

The deviation chord height is the maximum distance between the geometry of the 3D model and the surface of the STL or 3MF file.

The recommended value for the chord height is 0.05 mm. A smaller deviation chord height will result in a more accurate surface.
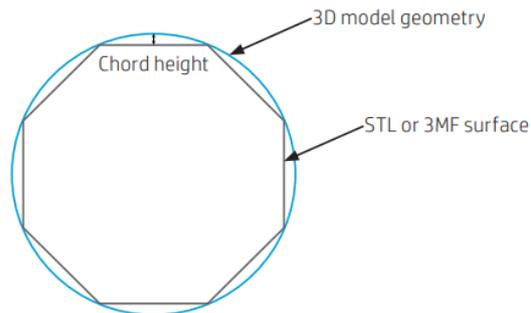

Figure 3. Deviation chord height

Angle Tolerance

The angle tolerance is the maximum angle between the normal vectors of adjacent triangles. The recommended value for the angle tolerance is 1 degree.
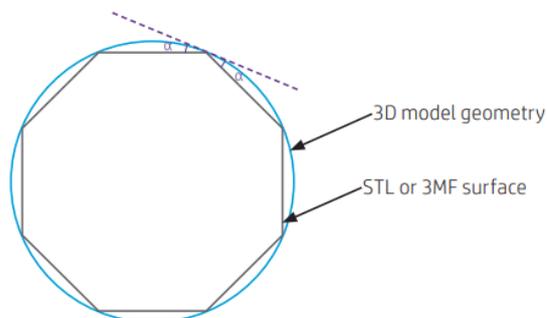
Figure 4. Angle tolerance

Exporting Errors

Unexpected results such as surface inaccuracy (e.g., unexpected holes, unjoined triangles, overlapped triangles, tiny shells, flipped-direction triangles) or poor resolution are common errors that may occur when an STL or 3MF file is inadequately exported.

Too Many or Too Few Triangles

Although a mesh with more triangles tends to be more accurate, too many triangles are difficult to process and, when a certain size is reached, the additional triangles do not provide enhanced accuracy. For this reason, an excess number of triangles could increase processing time with no benefit.
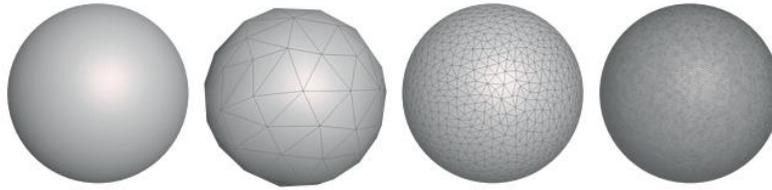
Figure 5. Examples of tessellation (from left to right): 3D model, tessellation with too few triangles, correct tessellation, tessellation with too many triangles

Similarly, too few triangles can lead to poor resolution results. Triangulation of a surface causes faceting of the 3D model.

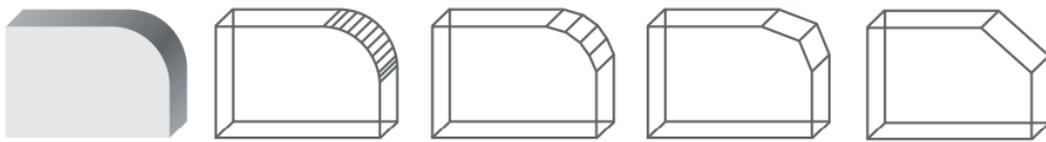The exporting parameters used to output an STL or 3MF file affect how much faceting occurs.

Figure 6. Exporting quality is dependent upon 3D model design

A small mesh doesn't necessarily mean lower quality. Rectilinear surfaces result in very high quality with a very small number of triangles. The necessary exporting quality is very design-dependent.

Repairing STL Files

Common errors can normally be fixed by properly designing and exporting the 3D model using CAD software or another appropriate repair software. The most common software for repairing STL or 3MF files are the following:
• Materialise Magics with Materialise Build Processor
• Autodesk® Netfabb® Engine
• HP SmartStream 3D Build Manager

Dimensions in Printer Resolution

The minimum controllable printable volume when printing with HP Multi Jet Fusion technology is known as a voxel, which
defines the resolution.

The HP Multi Jet Fusion voxel resolution in the Z axis is 80 microns. Thus, it is important to align critical dimensions to an integral number of voxels: It is possible to obtain a 3D printed block of 160 microns or 240 microns, but it is not possible to obtain one with 168.5 microns.
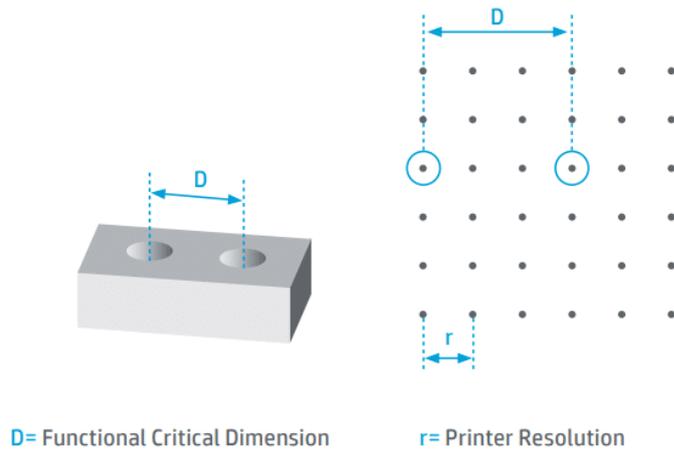
D= Functional Critical Dimension   r= Printer Resolution

Figure 7. Printer resolution