DESIGN GUIDE

HP Multi Jet Fusion

WHERE **BIG IDEAS** ARE **BUILT**

Table of Contents

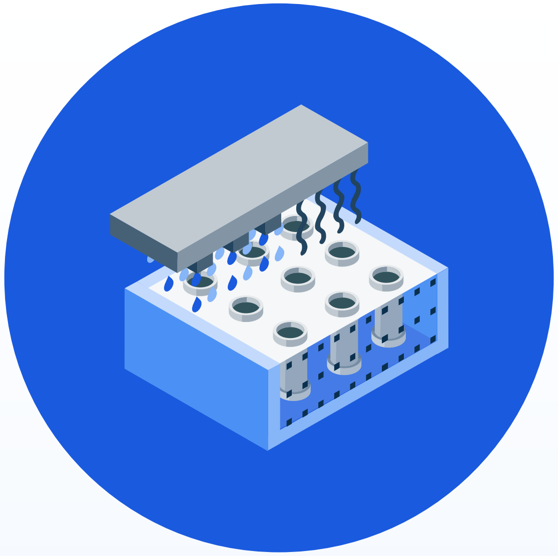
MULTI JET FUSION

Introduction

What is Multi Jet Fusion?

HP's Multi Jet Fusion is a unique 3D printing process vastly different from the others on offer at Xometry. However, it is no less precise and can create high-quality parts up to 10X faster than competing 3D printing processes, allowing you to get to market faster.

Like all powder-based 3D printing processes, HP Multi Jet Fusion (MJF) builds parts layer by layer. MJF uses fusing and detail agents, deposited on part cross sections per layer in a method similar to ink-jet printing. The cross sections are then exposed to intense heat, fusing the cross section and the layer underneath. This process is rapidly done to set each layer before moving on to the next. In more traditional 3D printing processes — such as Selective Laser Sintering (SLS), Stereolithography (SLA), or Direct Metal Laser Sintering (DMLS) — each part layer is imaged with a single laser beam which make the time to print each layer variable based on the cross sections. MJF does not have this limitation, giving an even print time per layer, giving a throughput advantage over these other processes.



HP Multi Jet Fusion is an extremely exciting, robust, and flexible addition to Xometry's manufacturing capabilities. It's excellent for prototyping, small-batch production runs, or as a bridge process to injection molding. It lets you get a feel for how your parts will perform with minimal upfront costs and great economies of scale.

The Benefits of HP's Multi Jet Fusion 3D Printing Process

While there are many benefits to HP Multi Jet Fusion, a few truly stand out. For starters, the standard build parameters are optimized for the best density. The result is that MJF parts are water-resistant and can be fully watertight with post-processing operations like vapour smoothing (assuming that the design of the part is right and that the manufacturer has the necessary capabilities).

Multi Jet Fusion is the way to go if you want to 3D print at higher quantities for small-batch production runs. The ability to print multiple parts simultaneously across the entire build volume means you can print parts at rates up to 10X faster than SLS or other 3D printing processes. Multi Jet Fusion also delivers more balanced mechanical properties across the X, Y, and Z axes than similar processes, such as SLS.

MJF also does not require support structures for printing parts, allowing for 3D nesting of parts in the build area for bulk processing. Parts fused in MJF do not require support because they are suspended in unfused powder throughout the build. After building, parts are allowed to cool, removed from the loose powder, and bead blasted for cleaning before any secondary processing.

If you're interested in injection molding for your project, getting a 3D printed part for testing form and fits is always a good idea before investing in metal molds. While SLA is a great 3D printing process for extremely detailed and high-resolution prints, the UV-cured resins are not as tough as traditional thermoplastics. Prints begin to degrade in UV light and moisture. On the other hand, Multi Jet Fusion can produce extremely accurate prints while maintaining the structural durability of traditional thermoplastics. This makes it a great process for testing fit and functionality before taking your project to injection molding.

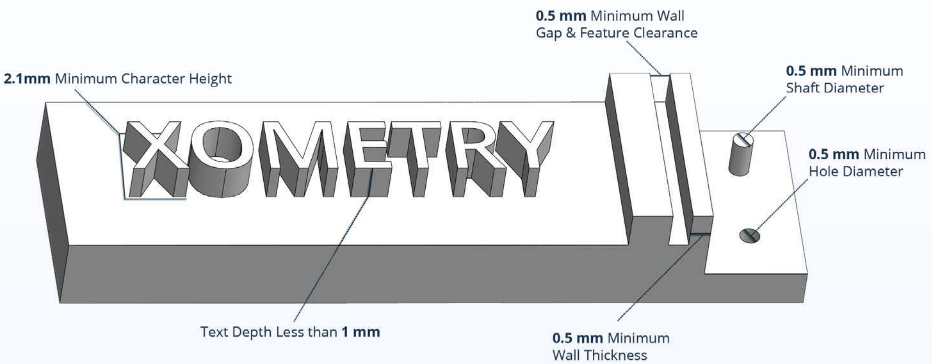
MULTI JET FUSION

Multi Jet Fusion Design Guidelines and Best Practices

Features and Sizes

There are some very important design specifications to bear in mind to avoid issues in the printing process and to achieve the highest part quality possible. The minimum printable features in the X, Y, and Z planes:

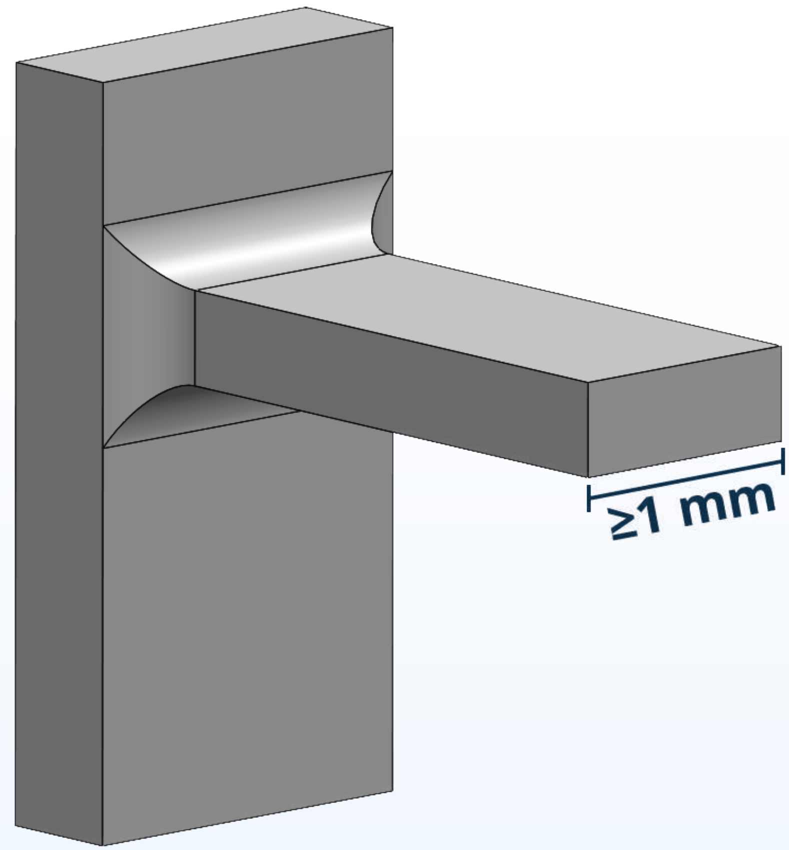
Feature	Minimum	Recommended
Wall thickness	0.5mm*	1.5mm
Hole diameter at 1 mm thickness	0.5mm	1.5mm
Shaft diameter at 1 mm height	0.5mm	1.5mm
Clearance at 1 mm thickness	0.5mm	1mm
Gap size between walls	0.5mm	1mm
Character height for embossed or debossed text	2.1mm	5mm
Debossed or embossed text depth	1mm	Greater than 1mm**
Feature thickness	0.5mm	Less than 4mm



* For bulky parts, the maximum wall thickness is 7 mm. If the wall thickness is higher, The parts must be hollowed.
** Text depth can exceed 1mm; however, the greater the depth, the more challenging it becomes to remove powder from the crevices of the text.
The shallower the text, the easier it is to post-process, resulting in a crisp result.

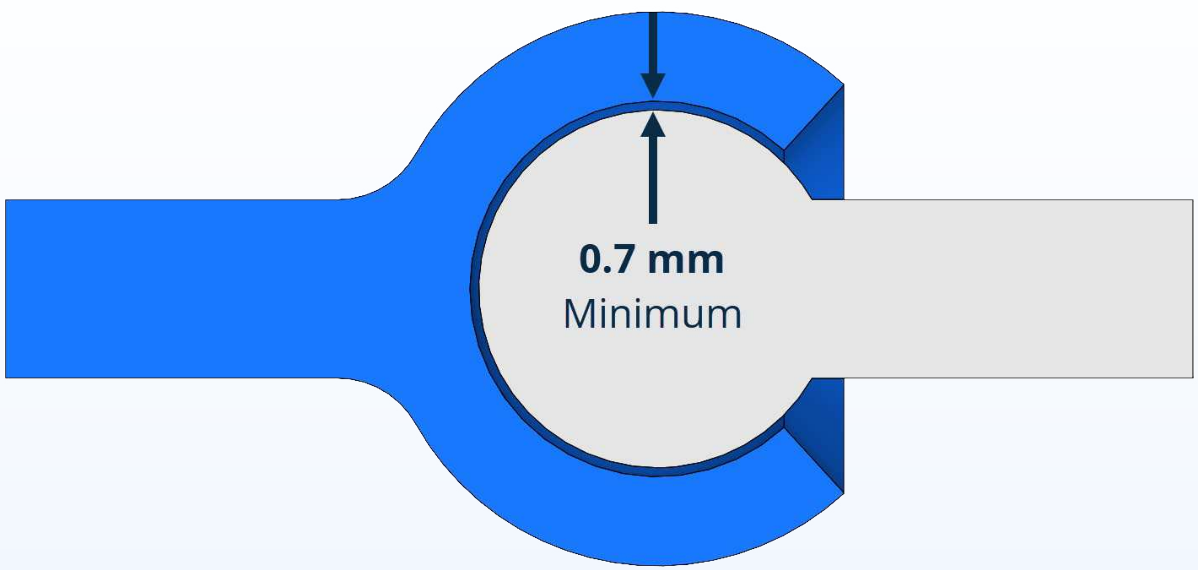
Cantilevers

A cantilever is a structural feature that extends outwards horizontally and is unsupported at one end. We recommend keeping the width of cantilevers above **1 mm**. For cantilevers with high aspect ratios (Length/Width), consider adding supporting ribs and fillets or increasing the wall thickness to strengthen the feature.



Assemblies

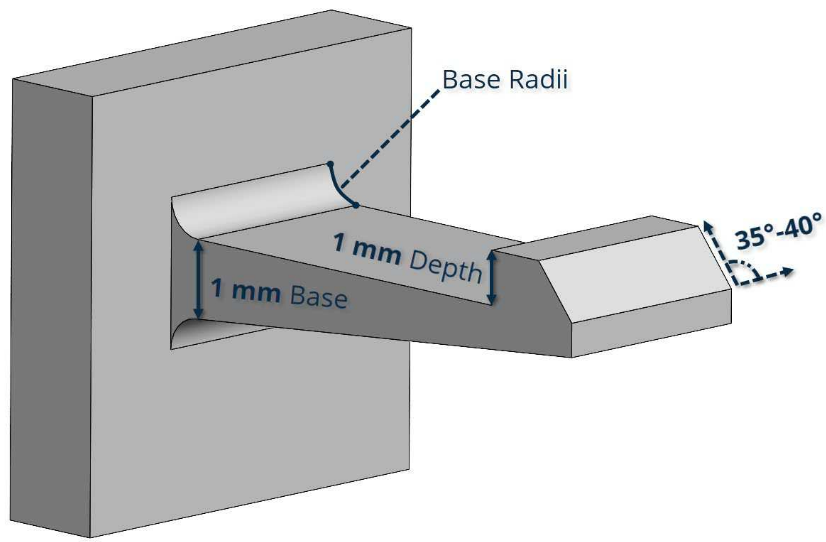
The minimum gap between assembled parts after printing should be at least **0.4 mm (± 0.2 mm of tolerance for each part)** to ensure correct assembly. If you want to do print-in-place assemblies such as a ball joint or hinge, we recommend a minimum clearance of **0.7 mm** between the parts to ensure movement. Parts with thick walls **above 30 mm** should have an even greater gap to ensure proper performance.



Pro Tip: Make note of floating or moving bodies within an assembly via your order notes or with an attached drawing. This highlights the areas for technicians during post-processing, which helps ensure powder removal and assembly functionality.

Snap-Fits

A simple and inexpensive way to assemble plastic parts together is with snap-fit features. These features attach parts together via a protrusion on one part that fits into a groove or slot on another. The protruding feature deflects during assembly and snaps back to its original shape after assembly, causing the pieces to become locked together. The most commonly used snap-fit is of the cantilever type.



We recommend following these guidelines when designing snap-fits for MJF 3D printing:

- Use a cantilever base thickness of at least **1 mm**
- Overhang depth should be at least **1 mm**
- Add radii half the size of the thickness at the base of the cantilever
- Add a chamfer to the end of the overhang to avoid sharp edges and breakage
- Keep the overhang assembly angle **between 35º and 40º**
- Taper the cantilever beam for more evenly distributed strain, reduced stress concentrations, and to lower the required assembly force

Warp Mitigation

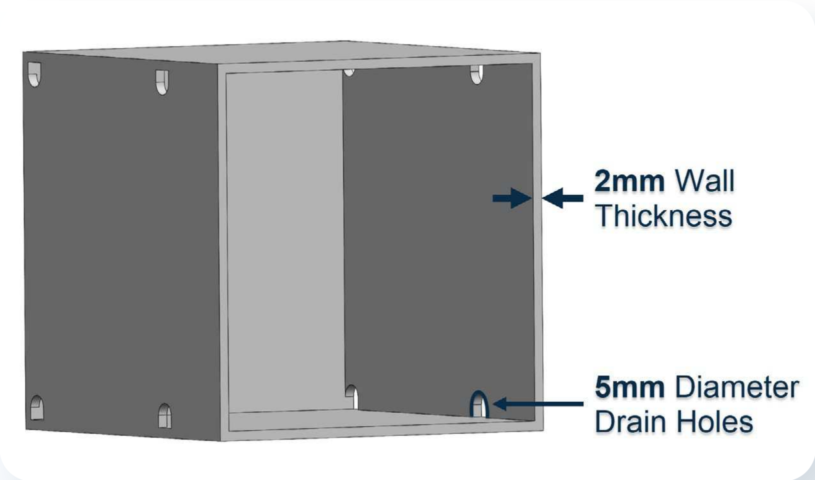
As MJF parts cool during printing, they experience a small amount of shrinkage. When one area of a part cools and shrinks before another, it causes stresses and tension that can lead to warpage and distortion. Parts especially prone to shrinkage-induced warping include thin and long parts, parts with broad features, parts with abrupt changes in their cross-sections, and parts with thin curved surfaces. Generally, a part with a length-to-width ratio higher than 10:1 is susceptible to distortion. By focusing our design around even cooling, warpage can be mitigated. Here are some techniques to improve cooling uniformity in your parts:

- Avoid ridges and ribs on broad areas
- Use smooth transitions in areas where significant cross-sectional changes occur
- Reduce aspect ratios of long walls by increasing their thickness
- Use lattices or hollowing to lighten and balance out thick areas

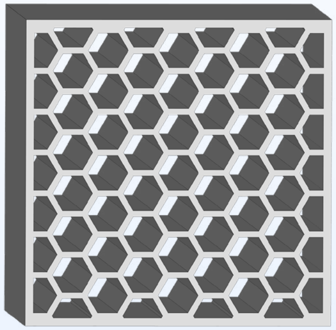
Hollowing and Structural Fills

We recommend hollowing out the part as much as possible when designing parts. This will save on fusing agent and powder, reduce sink marks, and reduce print time, saving you money. If your part is hollow and has a closed geometry, drain holes need to be added to the design to aid in removing the material. We recommend placing at least two drain holes on opposing faces of the part and using a drain hole diameter of at least **5 mm** or greater. When hollowing a part, we recommend a minimum wall thickness of **2 mm**. If your application demands it, increasing the wall thickness further can strengthen the hollowed part.

Another way to greatly reduce mass and material usage is by replacing solid areas with structural fill, such as honeycombs or lattice structures. These structures typically comprise repeating geometric patterns that create a network of rigid cells that help retain mechanical integrity while significantly reducing material usage. Professional 3D printing software such as Materialise Magics or nTopology can help automate replacing solid areas with lattices in your desi



Hollowed part with drain holes.

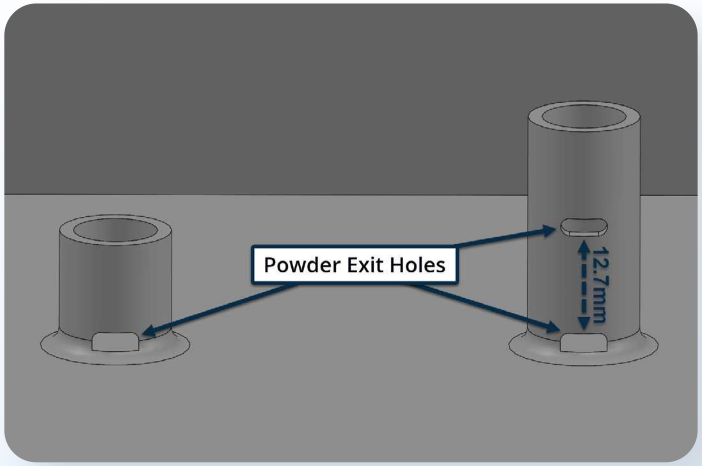


Part with a honeycomb structural fill.

Pro Tip: Place drain holes in sharp corners. Doing so ensures complete removal of excess powder during post processing and prevents build-up in corners caused by eddy currents.

Blind Holes and Bosses

When designing a part for MJF, one should always consider powder removal. In this regard, screw bosses and blind holes are often overlooked areas. Without an escape route, powder tends to accumulate at the bottom of these features. The deeper they are, the more likely it is for buildup to occur, making it increasingly difficult to remove the powder. A straightforward solution to this problem is to add exit holes or channels with a clear line of sight at the bottom of the hole, providing an avenue for excess material to exit. Depending its depth, you may need to add multiple exit points along the feature to ensure complete powder removal. We recommend placing exit points every **12.7 mm** for deeper holes.



Pro Tip: Add fillets to the base of bosses to increase structural integrity. The larger the fillet, the more surface area it will have and the stronger the connection will be.

Limitations to Consider When Printing With Multi Jet Fusion

Limitations to Consider When Printing With Multi Jet Fusion

Like all 3D printing processes, there are some limitations to consider before selecting HP Multi Jet Fusion as your additive manufacturing process. Multi Jet Fusion might not be the most cost-effective option if you're printing just a single, one-off part. The cost-savings for Multi Jet Fusion truly come into play when printing in higher quantities that can fully utilize the available build space.

HP MJF prints in a natural gray color with a slightly textured finish similar to SLS. The natural gray color may be somewhat inconsistent across a single part. We recommend a dyed black finish for the best consistency and cosmetic appeal.

Tiny or large parts that take up nearly the entire build volume may experience issues. Parts with thicker geometries, flat or broad parts, and parts with uneven wall thicknesses may be prone to significant deviations or warp due to variable thermal shrinkage and stress. Though Multi Jet Fusion is an accurate process, the standard layer thickness is 80 microns with a minimum recommended feature size of 0.5 mm, and tiny features or small parts under 0.5 mm can be lost or won't print correctly.

MULTI JET FUSION

Technical Specifications and General Tolerances

Technical Specifications and General Tolerances

The charts below are a great reference to keep on hand when designing for HP's Multi Jet Fusion 3D printing process.

Material Options	Nylon 12 Glass-Filled Nylon 12 Nylon 11 TPU 88A Polypropylene
Standard Layer Thickness	80 microns
Build Volume	380 x 284 x 380 mm 356 x 280 x 356 mm is recommended useable area
Minimum Printable Features	0.5 mm in X, Y, and Z planes
Colors & Finishes	Natural Gray Dyed Black (recommended) Vapor Smoothed Custom

Click Here To Learn More

HP MJF Prototyping Tolerances

In the table below, you will find our standard MJF tolerances for prototype orders. These include one-off or first-time prints, non-production orders, and auto-quoted orders that have not received a manual engineering review.

Material	Under 30 mm	30 - 50 mm	50 - 80 mm	> 80 mm
Rigid Materials (Nylon, PP)	± 0.70 mm	± 0.85 mm	± 1.40	± 1.75%
Rubber-Like (TPU)	± 1.05 mm	± 1.35	± 1.80	± 2.25%

Typical MJF Tolerances Based on HP 5200-Series, Balanced Mode

Engineered & Production MJF Tolerances

Tolerances tighter than prototype tolerancing can be guaranteed after additional engineering review. This includes higher volume production part orders and orders manually optimized by our engineering teams, often after an initial prototyping run.

Material	Under 30 mm	30 - 50 mm	50 - 80 mm	> 80 mm
Rigid Materials	XY = ± 0.25 mm Z = ± 0.42 mm	XY = ± 0.30 mm Z = ± 0.50 mm	XY = ± 0.39 mm Z = ± 0.60 mm	XY = ± 0.5% Z = ± 0.75%
Rubber-Like (TPU)	XY = ± 0.60 mm Z = ± 1.05 mm	XY = ± 0.60 mm Z = ± 1.35 mm	XY = ± 0.60 mm Z = ± 1.80 mm	XY = ± 0.75% Z = ± 2.25%

Engineered & Production MJF Tolerances Based on HP 5200-Series, Balanced Mode

MULTI JET FUSION

Materials Available for HP Multi Jet Fusion

Materials Available for HP Multi Jet Fusion

Xometry's HP Multi Jet Fusion service offers five different material options for printing: Nylon 12 (PA 12), Glass-Filled Nylon 12 (PA 12 GB), Nylon 11 (PA 11), Polypropylene (PP), and TPU 88A (BASF Ultrasint™ TPU01).

- **Nylon 12** is a great general-purpose option with balanced mechanical properties and produces high-density parts. This material is great for complex assemblies, enclosures, and watertight applications. Parts printed in Nylon 12 will offer the highest cost savings compared to the other MJF material options.
- **Glass-Filled Nylon 12** is a 40% glass bead-filled thermoplastic that provides an affordable, high-strength material option. Ideal for applications requiring high stiffness, such as enclosures and housings, fixtures, and tooling.
- **Nylon 11** has optimal mechanical properties, high chemical resistance, ductility, and enhanced elongation-at-break. Its pliable nature and strength mean that Nylon 11 can withstand impact, making it an excellent choice for defense, sports, and medical prosthetic applications.
- **Polypropylene** is a thermoplastic polymer widely used in various industries. It is known for its high chemical and moisture resistance, lightweight nature, and excellent thermal and electrical insulation properties. Common applications include packaging materials, automotive components, medical devices, and consumer products like containers, bottles, and household goods.
- **TPU 88A (TPU01)** is a thermoplastic urethane rubber with high levels of flexibility and tear strength. The material exhibits rubber-like qualities — like a shoe sole — allowing for various end-use applications.

Technical Specifications for HP Multi Jet Fusion Materials

Name	Technical Name	Standard Color	Other Finishes	Shore Hardness	% Elongation at Break (XY, ZX)	MPa Tensile Strength (XY, Z)	°C Heat Deflection (HDT @66 PSI, @264 PSI)	°C Melting Point
Nylon 12	HP 3D High Reusability PA 12	Matte Gray	Dyed Black Vapor Smoothed Custom	80D	20%, 15%	48, 48 MPa	175°C, 95°C	187°C
Glass-Filled Nylon 12	HP 3D High Reusability PA 12 Glass Beads	Matte Gray	Dyed Black Vapor Smoothed Custom	82D	6.5%, 6.5%	30, 30 MPa	174°C, 114°C	186°C
Nylon 11	HP 3D High Reusability PA 11	Matte Gray	Dyed Black Vapor Smoothed Custom	80D	55%, 40%	52, 52 MPa	185°C, 54°C	202°C
Polypropylene	HP 3D High Reusability PP	Matte Gray	None	70D (est.)	20%, 20%	30, 30 MPa	100°C, 60°C	187°C
TPU 88A	BASF Ultrasint™ TPU01	Matte Gray	Dyed Black Vapor Smoothed Custom	88A	220%, 120%	9, 7 MPa	N/A*	120-150°C

* HDT is not suitable for flexible materials as they bend easily, causing values to vary widely.

MJF Material Selection Guide

HP Multi Jet Fusion Material Selection Guide

	Rigid Polymers				Elastomeric
	Nylon 12	Nylon 12 Glass-Filled	Nylon 11	Polypropylene	TPU88A
Stiffness	Good	Best	Good	Fair	Not Recommended
Impact Resistance	Fair	Not Recommended	Good	Fair	Best
Elongation / Ductility	Fair	Not Recommended	Good	Fair	Best
Dimensional Stability	Best	Good	Good	Fair	Fair
Level of Detail	Good	Good	Best	Fair	Fair
Flat Parts	Good	Best	Fair	Not Recommended	Fair
Temperature Resistance	Fair	Good	Not Recommended	Fair	Not Recommended
Chemical Resistance	Good	Good	Good	Best	Fair
Low Moisture Absorption	Not Recommended	Not Recommended	Not Recommended	Best	Fair
Lightweight	Good	Fair	Good	Best	Fair

Key

- Best
- Good
- Fair
- Not Recommended

Details & Tips on Vapor Smoothing

Details & Tips on Vapor Smoothing

Powder bed fusion processes like HP Multi Jet Fusion typically leave parts with a matte, sugar cube-like surface finish. Chemical vapor smoothing is a technology that smooths the surface of 3D prints and enhances mechanical performance via vaporized solvents, making 3D printed plastics more suitable than ever for low-volume production.

How Does Vapor Smoothing Work?

Unlike physical smoothing processes such as sanding or media tumbling, vapor smoothing does not remove material from the workpiece. Instead, a vapor finishing agent (FA 326) is introduced into a sealed 600x400x400 mm processing chamber, where the parts are racked and hung for maximum surface exposure. The vapor clings to the part surface, creating a controlled chemical melt. This melt reduces the surface peaks and valleys by liquifying and redistributing material and evening the surface. The melt also has the effect of enhancing shine.

Once the parts are finished, the chamber is heated to evacuate the finishing agent to a collection vat. No residue is left on the parts. Smoothed parts are then ready for shipping or another secondary process, such as dyeing.

Benefits of Vapor Smoothing

Vapor-smoothed parts have improved cosmetics and are comparable to injection molding surface finishes. It is important to note that chemical vapor smoothing re-distributes surface material to provide a sealed, smooth surface but does not create a polished surface; there will still be visible surface topology and minimal build lines.

Chemical vapor smoothing can significantly increase the viability of 3D prints in food processing, medical devices, and consumer products and is recommended for end-use applications in these industries. Notably, vapor-smoothed nylon 12 3D prints effectively reduce bacterial attachment and growth compared to standard nylon 12 3D prints. Studies have shown that vapor-smoothed HP MJF parts conform to and pass multiple biocompatibility and safety tests, such as cytotoxicity, Microbiological MRSA Bacteria, and Microbiological E. coli Bacteria tests.

Other benefits of chemical vapor smoothing include:

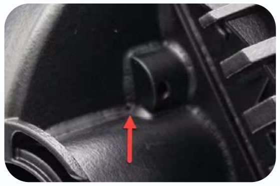
- A uniform, sealed surface on 3D printed parts
- Improved tensile, elongation, and flexural performance
- No extra coatings; the surface is the part's material
- Significantly reduced moisture absorption
- Closed loop, repeatable processing results
- Ability to smooth non-line-of-sight features
- Retains features and dimensional accuracy
- No residual chemicals or media in features
- Improved cosmetics and enhanced colors

Tips for Vapor Smoothed 3D Prints

While there are many benefits of chemical vapor smoothing, including smoothed layer lines, sealing micro gaps, and enhancing mechanical performance, the process can potentially introduce defects if your parts aren't designed accordingly. Below are the defects to be aware of and our tips for avoiding them in your designs.

Bridging - During the vapor smoothing process's liquefaction stage, a thin material shell may form between close features like small gaps or walls with sharp corners.

Design Tip: Increase the distance between gaps and add ample radii to sharp internal corners.



Bite Marks - "Bite marks" are a defect associated with the impression left from clips used to hold parts that do not have features that allow for hooking. These marks are usually more noticeable on softer materials like TPU.

Design Tip: Adding a hole or feature that can be used to hook through and rack the part instead of clamping will prevent bite marks from appearing on cosmetically important faces. Note which faces or features of your part are cosmetically important if you can't add a hooking feature, and our technicians will do their best to avoid clipping onto those areas.

Bubbles and Blistering - Similar to bridging, blistering results from the surface bubbling during the liquefaction stage. After solvent evacuation, the bubble may harden and leave a blister-like appearance.

Design Tip: Avoid broad, flat surfaces; organically shaped surfaces are less likely to blister. Adding radii on internal corners will also help mitigate bubbling.

Breakthrough and Holes - Unintentional holes may appear on very thin areas. This happens when the surface on each side of the thin wall has liquified to the point that the center has become fully liquid. This causes a hole to form when the material flows and re-hardens.

Design Tip: Keep walls at least 1mm thick to prevent the formation of holes. Maintaining uniform wall thickness throughout the part also helps prevent printing defects.

Additional Resources

Additional Resources at Xometry

Online Instant Quoting

- Web: Upload your CAD file at get.xometry.com.tr
- Accepted file types: STEP (.step, .stp), SOLIDWORKS (.sldprt), Mesh (.stl, .3mf), Parasolid (.x_t, .x_b), DXF (.dxf), Autodesk Inventor (.ipt), Dassault Systems (.3dxml, .catpart), PTC, Siemens (.prt), ACIS (.sat), and more!
- Other Xometry Capabilities:


CNC Machining

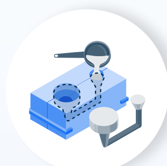

Sheet Metal Fabrication


Sheet Cutting


3D Printing


Injection Molding


Urethane Casting


Die Casting


and More!

Contact Information

- Email: info@xometry.com.tr
- Phone: +90 (212) 221 06 3

Other HP MJF Resources

- HP MJF Technology Overview
- HP MJF Design Guidelines
- MJF Deep Dive Webinar [Recording]
- Vapor Smoothing