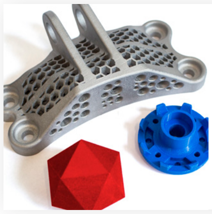
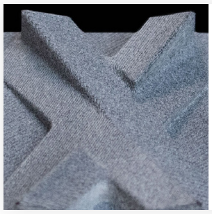
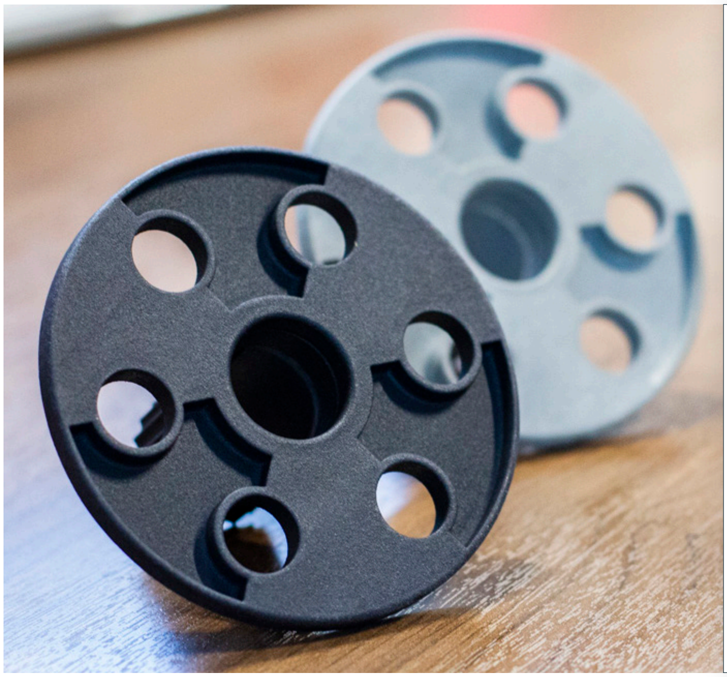
Xometry

The Ultimate Guide to 3D Printing

The Ultimate Guide to 3D Printing







Chapter 1

Introduction

What is 3D printing? Learn the basics of additive manufacturing.

Sections

Introduction

3D printing, or additive manufacturing, is an umbrella term for manufacturing technologies that can generate parts by growing them out of base material. This differs from subtractive manufacturing, such as CNC machining, where a bulk material is reduced to its final shape through cutting or forming.

3D printing is a powerful tool for generating custom parts, often with complex geometries, and serves industries from aerospace & defense, to automotive, to medical and dental, and beyond.

This guide outlines 3D printing, its basics, getting started, and the top commercial 3D printing processes and materials used today.

What is 3D Printing?

3D printing is a manufacturing process where materials are joined together to make objects from 3D model data (CAD). Typically, 3D printing is a layer-by-layer process where part geometries are "grown," fusing with the previous layer. 3D printing processes can build objects in plastics, photopolymers, reaction polymers, composites, metal, glass, and other materials.

This is contrary to subtractive, or traditional manufacturing, such as CNC machining, where a part is made through the reduction of a stock material through cutting, bending, shaping, etc. Due to the layered process of 3D printed parts, there are typically surface finish differences between prints and traditionally manufactured components. Most notably is a less smooth surface, with step-like features visible in the direction of growth. Another way to think about how 3D printing works is that it interprets a 3D model, acting as a digital twin, and does its best effort to reproduce the part features, building from the bottom to the top.

3D printed components, or those made on a 3D printing or additive manufacturing platform, are best known for their viability in small-batch manufacturing, also known as prototyping. This is due to the flexibility that 3D printing offers over traditional manufacturing, without requirements for specialized fixtures or labor-intensive setups. Over the last 5 years, there has been a spike in 3D printing used for end-use production components, from aircraft ducting to medical implants to commercial goods.

3D Printing vs. CNC Machining

3D printing differs from CNC machining because it fuses a smaller stock, such as a filament, powder, or resin, together to form a shape based on the interpretation of 3D data. CNC machining uses larger material stock, and cutting tools such as end mills and drills to reduce the materials to the final shape. 3D printing is typically a non-stop process from beginning to end, meaning that once the operator presses GO they would expect not to intervene until the printing process is complete.

CNC workflows require discrete operations, or “ops,” where a machining cycle is run to remove material that the tools can access, then often the part is moved to a different orientation and another op is conducted. Each setup for an op can add cost to a part, and with more complex parts at small volumes the price can get relatively expensive. With 3D printing, the entire part can be made at once with minimal post processing and no need for custom fixturing--being more cost effective for low volumes.

The freedom of complexity with 3D printing can be seen as a paradox against the expense it adds to CNC machining. In reverse, bulky items can be relatively cheap to machine compared to 3D printing, particularly with metals, due to the increased work a 3D printer must do to fuse the material together. Just like CNC machining requires design rules, 3D printing is not a cheat code for design either, it just has different strengths which can be powerful for lightweight, application-based, manufacturing.

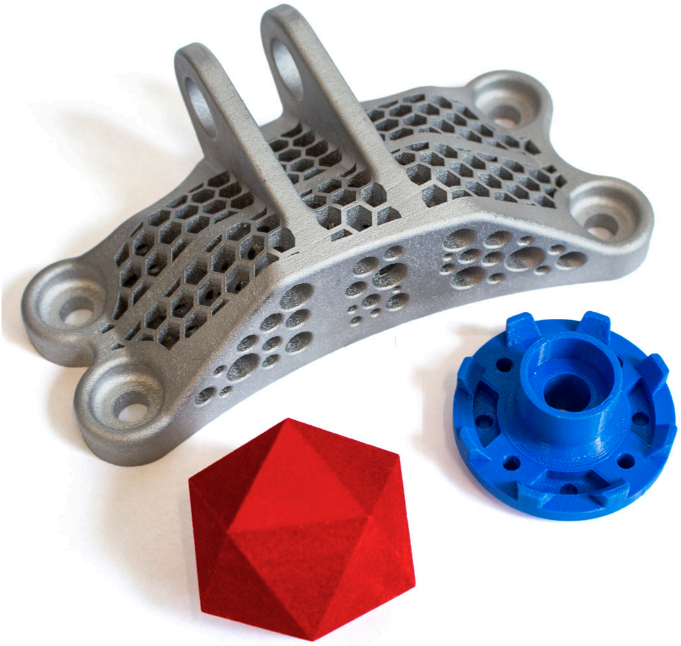
Injection molding typically requires a CNC machined mold core and cavity to run in an injection mold press. The machined plates are pressed together and a molten plastic is injected into the sealed cavity to cool into the shape of the part. Once cooled, the two halves of the mold open, and the part is ejected leaving the tool ready for the process to repeat. The major expense in injection molding is the upfront tooling required. Not only does it require a significant financial investment, it also often takes weeks to months to have the tool produced for a production run.

Because 3D printing can make parts directly from a digital file without tooling, it is a viable way to build production volumes of parts in quantities without significant investment. 3D printing often has a break-even between 250 and 2,000 parts before injection molding becomes a more viable option. With more lean inventory methods like just-in-time (JIT), 3D printing can be a replacement to molded components with limited on-hand needs.

3D printing can be used to build molds, or mold inserts used for prototype and bridge tooling. Although typically not as robust, a 3D printed mold can often withstand enough cycles to validate a molded project in its final material. 3D printing injection molds still requires the molding platform and often is best utilized by the molding business as a form of rapid validation or bridge tooling.

3D printing processes like stereolithography (SLA) have commonly been used for fit check validations before moving to more costly tooling. This is to mitigate risk of moving to a final production tool, requiring thousands to tens of thousands of dollars investment, and finding the revision needs to change after tooling is produced. Beyond a substitute for low volume production, 3D printing is also a crucial step to ensure the molded project will be successful.

3D printed parts will have a different surface finish and material properties compared to the more established injection mold process. Substitutes and trade offs are required for most projects that require a cosmetic smooth finish or specific material grade. Novel 3D printing processed like Carbon DLS are bridging the gap between mold-like surface finishes and end-use properties, but the choices available for injection molding are substantially more robust.



Chapter 2

How to Get Started With 3D Printing

What you need to get started and how to prep your files for printing.

Sections

- 3D CAD Models
- Understanding Support Material
- 3D Printing Services vs. Ownership

3D CAD Models

Preferred File Types

Most 3D printing services, like Xometry, accept major file formats. This includes:

- STEP (.step, .stp)
- Solidworks (sldprt)
- Mesh (.stl)
- Parasolid (.x_t, .x_b)
- Autodesk Inventor (.ipt)
- Dassault Systems (.3dxml, .catpart)
- PTC Creo, Siemens (.prt)
- ACIS (.sat)

STEP is a preferred CAD file format. Parasolid formats like STEP are universally accepted by most CAD and CAM platforms. Parasolids inherently hold unit information and other properties which takes the guesswork about size and scale versus mesh formats like STL and OBJ. Native exports from software such as Solidworks can also be helpful for services who may look at the parametric feature tree to troubleshoot issues with the model that could affect the print.

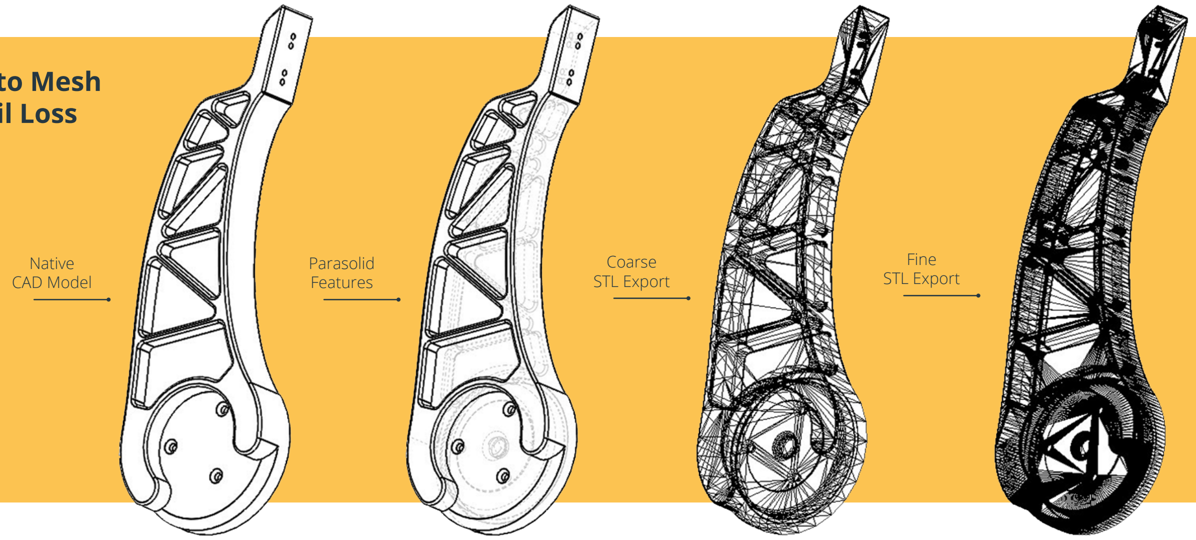
STL and Mesh Exports

Before being transferred to a 3D printer build setup software will require a mesh file, like an STL or OBJ. STL files can be directly uploaded or will be converted from the parasolid CAD file. STL files cannot be reversed back into parametric CAD. STL files hold mesh surface data as a series of triangles, so it is important to produce the file at a resolution that fits the 3D printing process while managing file size. The table below offers best practices for most STL exports.

STL Export Data	Best Practice
Mesh Resolution	0.01 - 0.03 mm
Chord Length	< 0.016 mm
Units	Millimeters (mm)

Mesh exports lose feature detail due to triangular tessellation of curved surfaces. This also makes reversing a STL to a parasolid difficult, if not impossible, without remaking the model.

CAD to Mesh Detail Loss



STL files do not contain any units or other part information beyond external mesh geometry. Because most 3D printers interpret files in millimeters (mm), it is important that the exported file is in millimeter units. If not, the file uploaded may be interpreted at a different physical size. For example, if a 1” part was exported to STL in inches, the STL would say the length is 1. When uploaded to a 3D printer, it would register a 1 millimeter--25.4 times smaller than it should be!

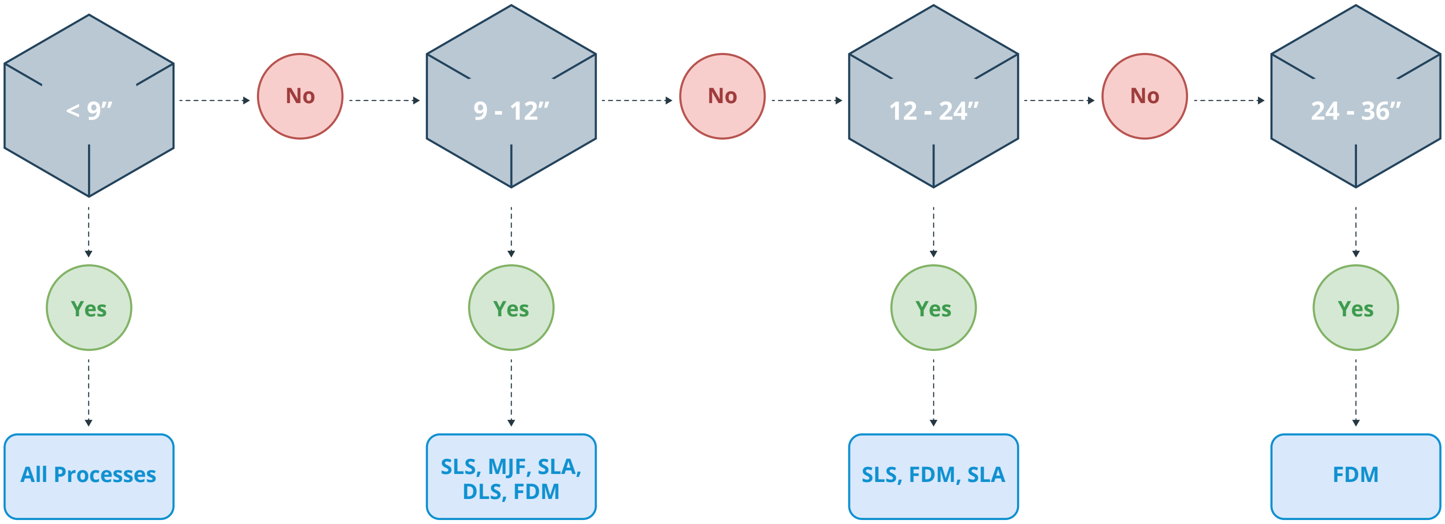
Design Best Practices

Each 3D printing process has its own specific design tips, which can be found in Xometry's detailed design guides at www.xometry.com. The tips below are great guidelines for most 3D printing processes.

Design Consideration	3D Printing Best Practice
Minimum Wall Thickness	It is recommended that features of any design are above 0.6 mm (0.024") on self-supporting features, like X-, T-, O-, or C-like shapes. At least 1.2 mm (0.048") is recommended for unsupported or load bearing features like I-like shapes, pins, or tabs.
Clearance Between Features	To ensure part features have a usable gap, it is important to design at least a 0.5 mm (0.020") gap.
Remove Confined Hollows	Most commercial 3D printing processes will have trapped material or support structure in confined hollows. Line-of-sight access to clean or drain these areas is necessary. These access areas are often called "escape holes," because they allow for material to escape during post processing. Any cavities with a depth beyond 50.8 mm (2.000") with only one access point, such as a boss, may require multiple escape holes for cleaning.
Fillet Everything	Fillets, or rounded internal and external corners, are preferred in every 3D printing process to mitigate transitions between features and reduce acute stress points. Generous filleting can increase the end-use performance of a printed part.
Be Mindful of Cantilevers	3D printing offers significant freedom of design. Features like cantilevers and "lollipop heads" can be delicate compared to the rest of the part. It is important to be mindful of these features while designing, especially if they are critical for function. It sometimes makes sense to make these features replaceable, or use off-the-shelf pins to produce these features on large parts where they can be crushed or damaged by the part's weight.
Design with Even Wall Thickness	3D printed parts share many design tips with injection molding, including uniform wall thicknesses. A uniform wall thickness will help mitigate any thermal deviations between thick and thin transitions on a part. These can be caused by uneven cooling of the part as it is growing or curing. Adding ribs, coring, and lattice structures can build strength without increasing thickness to a part. One exception is fused deposition modeling (FDM), where infills can be modified to sparse, lattice-like, cross sections which provide similar results.

3D Printing Build Sizes

Each 3D printing process has its own size limitations. The chart below quickly shows what sizes each platform can print. It is important to note that for most processes, building to the maximum size can increase risk to the part and the build due to increased time and layers required. A good rule of thumb is parts that fit on 1/6th the build area are more viable for repeated production.



Most processes can produce parts under 9 inches. Platform size can become a restriction for larger parts. Any parts over 36” must be sectioned for easier printing.

Reducing Costs for 3D Prints

Physical characteristics, materials selected, and machine platforms each play a role in the price of a 3D printed part. Many 3D printing service bureaus offer instant pricing, but may not be transparent on what the major cost drivers are. The list below provides a survey-level explanation of 3D printing price drivers.

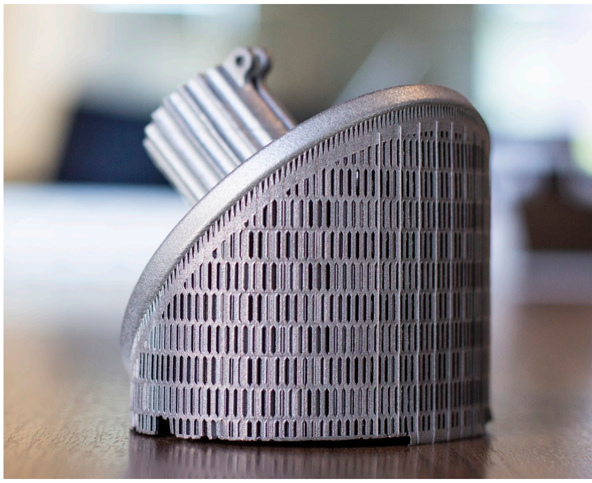
1) **The type of 3D printer** used can often be the highest cost driver for parts. Particularly in commercial additive manufacturing, machines have an overhead rate which incorporates the cost of ownership. Because many industrial platforms can cost hundreds of thousands of dollars, the rate can be between $15/hour for FDM platforms to over $100/hour for some metal 3D printing platforms. This hourly rate can be shared between parts, which mitigates the per unit cost and is one of the key benefits of bulk production with laser powder bed systems like SLS and HP MJF.

2) **The part geometry** is the second largest cost driver. Volume of material used, as well as build height (Z-height) can drive cost significantly. Z-height is less intuitive to a build than volume or area per layer because of mechanical operations that can happen every layer, or every several layers, on a machine platform. For example, many processes have a print head cleaning operation that runs periodically between layer movements. Although this may only take 10-30 seconds, over a large Z-height this can become an additional 20-30% build time.

Many systems also require mechanical movements, often adding 7-15 seconds per layer. Although this does not seem like too much it will aggregate significantly. If a part is 6” tall, and being ran at 0.005” layers (120 microns), then there will be over 1,200 layers in that build. If each layer change averaged 10 seconds, then over 3 hours of building would be consumed by these operations.

3) **The material chosen** is the third-largest cost driver. Commodity materials like ABS, ASA, and nylon often are the least expensive options due to their price and the platforms they run on. However, premium materials like Ultem or metals can sometimes add an order of magnitude to part costing due to material price as well as more specialized handling requirements.

Understanding Support Material



Support structure is unique to 3D printing and is generated along with the part to act as a base for overhanging features and is removed after the print is completed. In many 3D printing processes, the support structure is the same material as the part body. Some processes have secondary material which is sometimes soluble, or easier to remove from the part after printing.

Without support structure, an overhanging feature would not resolve or could have significant cosmetic issues. Support structure also plays a role as an anchor to the level build bed. Since it remains during printing, and after the print, support material can allow the 3D printed part to de-stress while maintaining its intended shape.

A few powder bed plastic processes, like Selective Laser Sintering, do not require support structure. Instead, the parts are suspended in their own build material.

Skipping ahead to the point: why not both? Additive manufacturing service bureaus offer a high variety of industrial manufacturing processes, a high amount of materials, and manage the supply chain for you--all with the touch of a button and easy-to-use interface. That being said, there are many times in product development where even waiting a few days can cause critical delays in product development. Personal desktop 3D printers can often generate a part in hours.

Personal 3D printers have become essential items for many engineering teams because they provide a “good enough” method of quickly iterating on a design, jig, fixture, or other development aid. Desktop platforms typically range from the low hundreds of dollars to the mid thousands, still much less expensive than commercial machines. Because the entry price is significantly lower, and reliability has increased with better software and hardware it often makes sense to have one or two machines on the office or shop floor. Ownership of a printer can also be a valuable training tool for engineers learning to design for additive manufacturing.

Ownership and services complement each other. Where in-house machines allow for rapid verification, they may not always offer the best materials, properties, build size, or capacity needed for projects. As a general rule, if an in-house machine will be occupied for more than 48 hours with a project, it is likely worth using a service bureau. Additionally, service bureaus have access to multiple platforms with larger build areas that can easily produce parts in days which could take a personal printer weeks to complete.

The more sophisticated the in-house printer is, the more likely staff and special work areas must be dedicated to run the machine. This can include increased labor, safety and personal protective equipment, hazardous materials disposal, and proper ventilation. Understanding the needs of the platform, the post processing, and the environmental requirements are important decisions on what 3D printer to own. Using 3D print services can help users decide which process would best suit in-house requirements.

Chapter 3

3D Printing Processes

An in-depth look at available 3D printing processes.

Sections

- Overview
- Material Extrusion
- Material Jetting
- Powder Bed Fusion
- Vat Photopolymerisation

Overview

3D printing, or additive manufacturing, has significantly advanced and diversified in the nearly 40 years since its invention. ASTM International, the global association which builds and defines industry standards, has created specific categories for different methods of 3D part manufacturing. The list below includes some of those categories, however novel processes and hybridizations are emerging every year, challenging the global definitions.

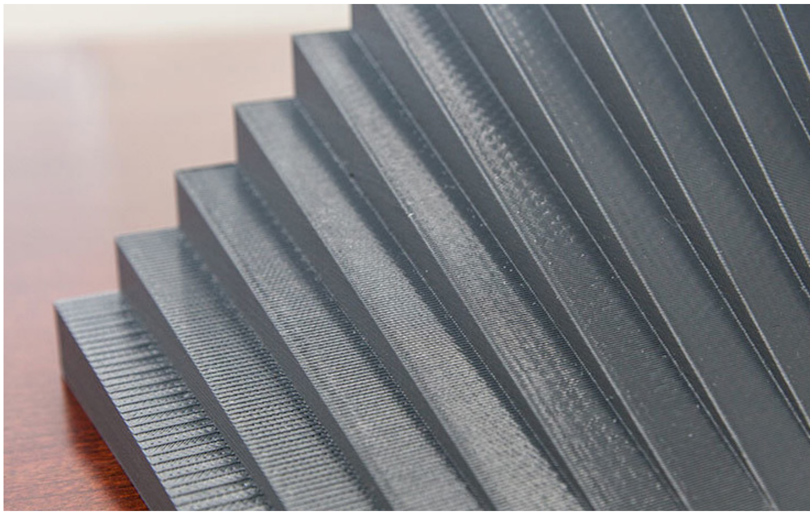
The current additive manufacturing categories, or families, are:

- Material Extrusion
- Material Jetting
- Powder Bed Fusion
- Vat Photopolymerisation
- Binder Jetting
- Directed Energy Deposition
- Sheet Lamination

Material extrusion is an additive manufacturing where a shaped bead of material, often melted, is extruded through a computer-controlled nozzle. This material builds on top of itself, the build platform, or support structures, to form the shape of the parts.

Fused Deposition Modeling

Fused Deposition Modeling, FDM, is also known as Fused Filament Fabrication, FFF, in its desktop counterparts. The FDM process uses a reel of thermoplastic-based material. The filament material is unspooled and fed into a heated extruder where it is deposited by a nozzle. For most FDM platforms, the nozzle moves in the X and Y directions while the platform moves further away in the Z direction as the part is formed.



FDM has a unique benefit of being user friendly because it does not need a sealed build chamber or complex thermal properties. The raw material is also easy to handle and store in an open environment. Because the material instantly hardens after being deposited, the FDM process requires support structure for overhanging features. This also allows various infills to be generated beyond a solid center to the part. Infills are used reduce material and time used to produce parts with large internal mass. Infills look like internal grids in the part. Although infills do provide benefits to weight and price, they may reduce the part’s strength versus a solid fill.

Commercially used platforms typically have FDM machines with at least two materials and extrusion heads--one for the part and the other for the support material. Support material is often soluble in sodium hydroxide (NaOH) or caustic solutions, leaving only the completed part after a soaking cycle. Professional 3D printing services like Xometry use the Stratasys Fortus FDM line due to its high reliability, diverse material selection, and build platforms up to three feet. The Fortus line offers the following FDM materials:

Fortus Standard Materials

- **ABS-M30** is A general purpose ABS which can use soluble support structure with six color options: black, blue, dark grey, ivory, red, and white.
- **ABSi** is a translucent ABS material which looks diffracted when grown due to the criss-crossing of layers in the process. ABSi has three color options: translucent natural, translucent orange, and translucent red.
- **ASA**, or acrylonitrile styrene acrylate boasts slightly improved performance over ABS as well as better color stability under UV exposure. ASA has ten color options: black, dark blue, dark gray, light gray, green, ivory, orange, red, and white.
- **Nylon 12** has higher impact strength and ductility than ABS, making it a better choice for ruggedized applications. FDM Nylon 12 is available in black.

Fortus Engineered Materials

- **ABS-ESD7** is an ABS formulated to be static dissipative. ABS-ESD7 is commonly used for ductwork, vents, electrical panels, and EMS fixtures. ABS-ESD7 is available in black.
- **ABS-M30i** is an ABS that can be sterilized using ethylene oxide (EtO) or gamma methods for ISO 10993 USP Class VI biocompatibility. ABS-M30i is available in ivory.
- **PC**, or polycarbonate, has a higher rigidity and heat deflection compared to ABS. Polycarbonate is available in a stark white.
- **PC-ABS**, is a blend of polycarbonate and ABS which balances the flexibility of ABS with a higher strength and heat resistance of polycarbonate. PC-ABS is available in black.
- **PC-ISO**, has similar properties as FDM PC and can be sterilized using ethylene oxide (EtO) or gamma methods for ISO 10993 USP Class VI biocompatibility. PC-ISO is available in translucent natural as well as white.
- **PPSF**, or polyphenylsulfone, is a highly stiff and brittle material with a significantly high melting point at 230 Celsius. PPSF is typically used for laboratory fixtures due to its resistance to several processing chemicals. PPSF is available in a natural beige.
- **Ultem 9085**, is a polyetherimide (PEI) thermoplastic that is flame retardant and offers the highest performance compared to other FDM materials. This included a high tensile strength, impact strength, and heat deflection. Ultem 9085 is available in black and natural tan.
- **Ultem 1010**, is a polyetherimide (PEI) thermoplastic that is flame retardant and has even higher tensile strength and heat deflection than Ultem 9085. Ultem 1010 can be steam autoclaved for sterilization. Ultem 1010 is available in amber.

Material jetting is a process where microdroplets of material are selectively deposited on a build platform. Looking similar to a 3-dimensional inkjet printing process, material jetting can produce high detail parts and often boasts the ability to use multiple materials, multiple colors, or both. It performs this by selectively depositing a material pixel-by-pixel.

This is not to be confused with binder jetting, where jetted binder is used to hold together a material, layer by layer, typically on a powder bed. Material jetting is depositing the part material directly.

PolyJet

Polyjet, a portmanteau of "polymer jetting," is a material jetting process that selectively deposits a UV-cured liquid resin using ink-jetting. A lamp on the printhead cures the deposited materials in a single motion, hardening the liquid, and the build tray is incrementally moved away from the print head to build height.

PolyJet requires support structure which deposits in the same method, looking like a gel. PolyJet support structure is usually removed manually to preserve fine details, and then finished with a water jet or soluble bath as a finishing step. Because support structure is another material being jetted, there are often indistinct differences on the outside surface of the part which has a tacky feel unless thoroughly treated or scrubbed with an abrasive, like a scotchbrite pad.

PolyJet materials include:

- VeroWhitePlus RGD835
- VeroBlackPlus RGD875
- VeroGray RGD850
- VeroYellow RGD836
- VeroCyan RGD84
- VeroMagenta RGD85
- VeroBlue RGD84
- Green - RGD5160-DM
- Ivory - RGD5160-DM
- High Temp RGD525
- VeroClear RGD810
- RGD720
- DurusWhite RGD430
- Endur RGD450
- TangoPlus FLX930
- TangoBlackPlus FLX980
- TangoBlack FLX973
- TangoGray FLX 950

Digital Materials

A unique trait of material jetting is the ability to take individual materials and deposit them in a microscopic digital matrix to modify the overall part properties. For example, PolyJet's Vero line are rigid, plastic-like, photopolymers and its Tango line are rubber-like materials. Tago materials inherently have a low Shore A durometer, but a digital material incorporating some Vero rigid materials in the matrix can simulate higher Shore A values, and even lower Shore D values without adjusting the feed materials in the printer.

Digital materials can also be used to combine pigmented resins together to print colors, images, and textures in three dimensions on a part. Combined with the ability to print with varying rigidity and durometer, material jetting platforms are powerful engineering prototype platforms for engineering and marketing evaluations.

Digital combinations in Polyjet are pre defined by software (prefixed DM-) and typically exhibit mechanical properties between rigid and flexible materials.

Multi-Material and Simulated Overmolding

Digital materials can be selectively applied to digital bodies of the 3D model. This can be used to simulate overmolds, often where a rubber feature is on a rigid substrate, such as a hand grip without requiring separate prints. In some manufacturing technologies, this can also be used to build supporting or sacrificial structures--such as XJet's technology which can print metal and ceramics through material jetting.

Powder Bed Fusion

Powder Bed Fusion, or PBF, is an additive manufacturing process where plastic or metal parts are formed by selectively melting model features in powdered raw material. Typically, this process is done layer by layer fusing parts with a heat source such as a laser. Because PBF creates fully dense parts in real metals and thermoplastics, it is widely used for end-use production components.

Additionally, plastic PBF has the benefit of not requiring sacrificial support structure. This means that parts can be bulk nested in a build using all available build volume. Printed thermoplastic parts using PBS processes are suspended in loose, unfused, powder which is typically removed by bead blast during post processing. Because of these benefits, PBF plastic processes like Selective Laser Sintering and HP Multi Jet Fusion tend to be the most economical methods of quickly producing end-use 3D printed plastic parts.

Complexity, Lattices, and Digital Foams

A common theme for many PBF manufacturing technologies is their ability to accurately produce complex organic features. These features are often so complex that they require sophisticated 3D computer generation software, such as that used in topology optimization or lattice generation. Lattices, in particular, are a strong suit of the PBF process because the process is typically more forgiving when resolving fine features and the materials themselves are more robust. Lattice structures can produce strong structure features without unnecessary weight; a powerful application for aerospace and medical applications.

In some thermoplastic PBF platforms, such as Selective Laser Sintering, the use of lattices combined with the ductile materials can create “digital foams.” These digital foams can be tuned to absorb impact and distribute energy with a 3D printable, open-cell design. Digital foams can substitute for traditional uses of foam in padding, insoles, sportswear, and other lifestyle industries. Because of the open structure of a lattice, digital foams offer better ventilation and heat distribution versus closed cell poured, casted, or molded foams.

Selective Laser Sintering

Selective Laser Sintering, widely known as SLS, is a laser powder bed fusion process (LPBF) where a fine layer of nylon powder is spread across a heated chamber and a laser will fuse cross sections of the part. Once a fully layer of cross sections are fused, both in the XY and Z direction, fusing that layer to the part features already created, the build area moves incrementally down and a new layer is deposited for the process to repeat. The major players in industrial SLS 3D printing equipment are EOS (Electro Optical Systems) and 3D Systems. industrial competition is emerging, such as Farsoon, with the expiration of key patents as well as equipment from FormLabs and others with lower-cost alternatives.

SLS is a powerful way of making plastic parts in bulk due to no support structure requirements. The fusion process, known as sintering, creates solid parts with a matte, sugar-cube-like surface finish. Typically SLS parts are made in nylon--most notably nylon 12 (polyamide 12, PA12) due to a very narrow melt temperature window and high viscosity, leading to better control and feature resolution on the parts. The SLS process can also produce parts in nylon 11 (polyamide 11, PA11), TPU, PS, PP, PEKK, PEEK, and others. Variants of nylon 12 and nylon 11 are by far the most commercially available.

Some SLS materials have been enhanced and modified with different filler options such as glass bead, mineral, carbon, and flame retardant additives. It is important to note that additives like carbon fiber are not long strand due to the powder state of the raw material, however they can help make parts lighter and stiffer at the cost of part ductility.

Selective Laser Sintering (SLS) 3D Printed Materials

Material Class	Desciption
Unfilled Nylon 12	General purpose, ductile material. Typically white.
Glass-Filled Nylon	General purpose, rigid material. Typically light beige.
Carbon-Filled Nylon	Light, stiff performance material. Typically very dark gray or black.
Aluminum-Filled Nylon	General purpose, rigid material. Typically light gray.
Unfilled Nylon 11	Highly ductile plastic. Typically white.
Flame Retardant Nylon	UL 94 HB and V-0 passed. Typically white.
Polypropylene (PP)	Chemical and moisture resistant. Typically natural.
Polystyrene (PS)	Delicate, low-ash, material for casting.
PEEK / PEKK	High performance, moisture resistant, heat resistant. Typically beige.

HP Multi Jet Fusion

HP Multi Jet Fusion (MJF) is a 3D printing powder bed fusion process developed by Hewlett Packard. Like Selective Laser Sintering (SLS), this process starts with a heated powder bed of thermoplastic material. However, HP MJF does not use a laser to fuse parts like SLS. The MJF platform uses an ink-jetted fusing agent which will specifically deposit on the part cross sections for that build layer. A second material, called a detailing agent, is also deposited around the edge of the fusing agent to create a defined edge for the part slice. This is done quickly in one motion, and is followed by a heat bar which travels across the entire build area.

The heat emitted is not enough to melt the untouched material, but does cause a melt where the fusing agent is present creating a solid part feature as well as fusing to the part layer underneath. Once the heat bar passes, the build area is lowered slightly and a layer of fresh powder is smoothly deposited for the process to repeat.

Like SLS, HP MJF can build parts without support structures as they are suspended in unfused powder during the build. The parts are later separated from the powder during post processing and bead blasted to remove excess material. A specific advantage HP MJF has over SLS is the time it takes to fuse each layer. SLS will take more time per layer for larger cross sections, which will vary each build layer. For the HP MJF system, the fusion per layer is much more consistent due to the ink-jetting depositing fusing and detailing agent simultaneously as it travels quickly across the build area, leaving an unchanging time between layers for fusion.

Because the processes is similar to SLS, many of the material options are nylon (polyamide, PA) based. Part properties overall are similar to SLS with slightly higher fatigue resistance. Due to the fusing and detailing agent, HP MJF parts will look matte grey on the exterior and be jet black on the interior. Below are some of the common materials available for HP MJF.

HP Multi Jet Fusion 3D Printing Materials Include:

- **HP 3D High Reusability PA 12** builds parts in general purpose Nylon 12. This is the most widely available material for the process. PA 12 is a good blend of stiffness and flexibility that is useful for housings, fixtures, and ductwork.
- **HP 3D High Reusability PA 12** Glass Beads is another common material offering 30% glass fill for increased stiffness compared to PA 12. The material is more dimensionally stable and less prone to warp and deflection.
- **HP 3D High Reusability PA 11** is a nylon 11 that is more ductile than nylon 12. PA 11 is useful for snaps, clips, and features that may require higher impact resistance.
- **BASF Ultrasint TPU01** is a thermoplastic urethane material with rubber-like flexibility. TPU is useful for wearables, digital foams, insoles, and other features requiring a tough, rubber-like material.

Direct Metal Laser Sintering

Direct Metal Laser Sintering (DMLS), is a metal additive manufacturing process where powdered material is fused by a laser on a layer-by-layer basis creating a dense metal object. Other names for DMLS are Direct Metal Laser Melting (DMLM), LaserCusing, and Selective Laser Melting (SLM). Like other powder bed fusion (PBF) processes, DMLS will selectively melt and fuse a cross section of the 3D printed part each layer, with enough energy that the melted material also fuses with the layer(s) underneath. Unlike the plastic PBF technologies, such as Selective Laser Sintering, DMLS requires the first layer to be fused with the build plate as well as sacrificial support structures to be generated due to intense stresses created when melting and cooling metal. This means that 3D printed metal parts in DMLS can be nested beside each other, but not above each other like plastic powder bed technologies.

Metal 3D printed parts via DMLS will have a matte grain finish, similar to cast parts. This is due to the base material being a powder and how the laser fuses the features when building the part. Supported faces will often show small dimples or a grid-like pattern where sacrificial structures were removed. 3D printing metal platforms include those from industrial OEMs like EOS, SLM Solutions, and Concept Laser.

Direct metal 3D printing materials consist of:

- **Iron-based alloys** such as 316L stainless steel, 17-4PH stainless steel, and maraging steel used for industrial, automotive, and medical applications.
- **Aluminum alloy** like AlSi10Mg is very popular for stiff, lightweight, 3D prints and a direct substitute to cast aluminum components.
- **Titanium-based alloys** like Ti6Al4V Grade 5 titanium and ELI titanium used in aerospace and medical applications.
- **Cobalt-based alloy** like cobalt chrome molybdenum (CoCrMo) are used for medical devices.
- **Copper-based alloys** such as CuCr1Zr alloy which offers design freedom for thermal management devices and electrical connectors.
- **Nickel-based alloys** are widely used in aerospace due to their performance under high heat. Inconel 718 and inconel 625 are two common nickel alloys used in DMLS.

Vat Photopolymerisation is one of the longest established additive manufacturing processes in the industry. The process consists of a material chamber with a liquid material that is selectively cured using ultraviolet (UV) light. Typically this is a UV laser or a UV DLP projector. The build plate moves away from the curing surface and resin is either introduced by a recoater blade or the resin may naturally flow into place for the next layer. Due to the specific nature of curing, the materials used in vat photopolymerisation are thermoset photopolymers. These materials have been specifically engineered to act “like” different common thermoplastics like ABS, polypropylene, or polycarbonate. After the initial print, most 3D printed photopolymers require a secondary ultraviolet cure in a specialized chamber. Some materials can also require post thermal cure to activate latent properties in the materials for increased performance.

Vat Photopolymerization processes, like Stereolithography (SLA), are known for achieving incredibly high detail in the prints as well as smoother surfaces versus other processes like Selective Laser Sintering (SLS) or Fused Deposition Modeling (FDM). Like FDM and DMLS, this process requires the use of sacrificial support structures to produce overhanging features. This requires the first layer of parts printed to be fused to the build plate.

Build Approaches in Vat Photopolymerization

Although a similar result, there are two main ways of growing 3D printed parts using vat photopolymerization. The first is with the build plate starting at the top of the resin bath, and moving downwards, submerging into the resin as the part is being cured on each respective top layer. In this method, the liquid resin is spread with a recoater blade to create a smooth surface before UV beam exposure. The UV laser or DLP projection is above the resin, beaming downward onto the surface. Each layer cycle hardens the resin on the layer beneath. This is often found on older platforms as well as large format machines.

Another approach is for the build tray to begin submerged in the resin vat and then progressively move up and above the resin as layers are cured. In this case, the part is built upside-down. In order to get a UV-cure, the resin basin must have a transparent bottom allowing a laser or projection to pass through. One benefit of this method is that less resin is required for a print, only needing enough to fully cover the bottom of the basin for an even cure.

This method also does not require a recoater, and often relies on natural liquid flow to fill gaps created as the build tray moves upwards. Some of these systems have incorporated a rocking mechanism to promote material flow as well as help release any hardened build material sticking to the transparent bottom of the resin basin. Parts built in this method may also benefit from more reclaimed material dripping from the part back into the build area. These types of platforms are often smaller size, however larger platforms are emerging using this method.

Stereolithography

Stereolithography, most commonly referred to as SLA, is a common 3D printing process using vat photopolymerization to create objects. SLA parts are formed by a ultraviolet (UV) light selectively curing cross-section profiles, layer-by-layer, from the bottom to the top. The UV light is typically from a UV laser, but can also be generated from a digital light projector (DLP). DLP methods benefit from being able to cure an entire cross section instantly, allowing for quicker times between layers.

SLA materials are typically engineered to act like traditional thermoplastics but are thermoset polymers. After printing, SLA materials go through a UV cure in a specialized light chamber to ensure all material is fully hardened. Sacrificial support structures are used in the SLA process to enable overhanging features. These supports are manually removed and residual features can be easily sanded.

Because SLA parts are built from a resin, their natural print surface is smoother than other 3D printing methods. SLA 3D prints benefit from easy post processing such as vapor blasting, sanding, polishing, or painting. This makes SLA materials ideal for cosmetic prototypes, model making, as well as fit check engineering models. Clear resins, such as Accura ClearVue, can be further post processed for near-optical clearly using manual polishing and glossy clear-coats.

Digital Light Synthesis

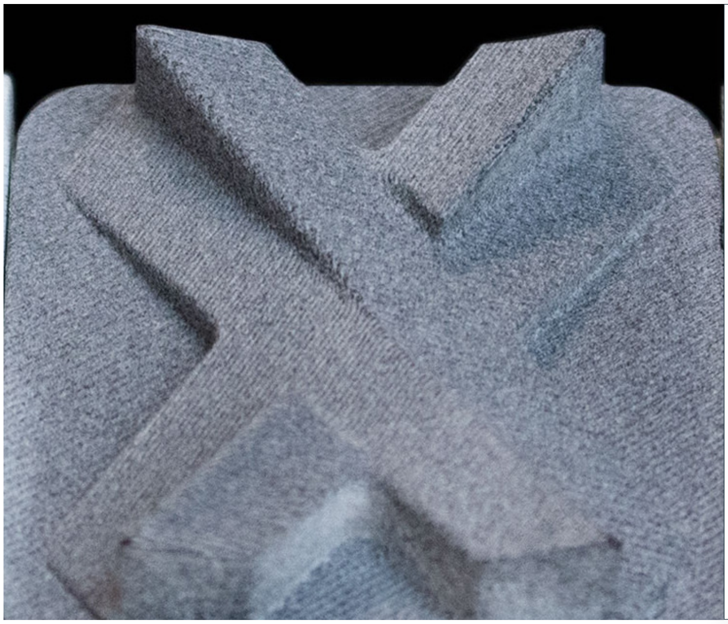
Carbon, Inc., has developed a process called Digital Light Synthesis™ or DLS™ for short. This process grows parts similar to DLP-based SLA methods, where the build tray begins submerged in a resin bath, and a DLP UV cross-section is projected through a transparent bottom of the resin basin. As the material is cured, the build plate lifts out of the basin with the parts.

Carbon DLS offers unique innovations for both the printing process and the materials used. The DLS process prints continuously, with the projector showing a video of part slide data corresponding with the Z movement of the build plate.

This is enabled due to a unique oxygen-permeable window on the bottom of the resin basin where the light shines through. This creates a very thin layer of material between the window and part which remains uncured even when exposed to UV light. The uncured liquid area readily flows between the cured materials above, and the window below during printing allowing continuous movement without a layer stop or any additional print release motions.

The second unique value proposition for Carbon DLS are the engineered materials used in the process. The continuous printing allows for feature sets to print isotropic, without a difference in mechanical ability dependent on build direction. For example, fused deposition modeling (FDM) is highly anisotropic, where the features built in the vertical direction are weaker than those built planar to the build plate. Most DLS materials require a UV secondary cure, similar to stereolithography (SLA), after printing and cleaning. Additionally, DLS materials can require a post-thermal cure through a controlled oven process to activate unique material chemistries enhancing the part’s physical properties.

Because of the end-use nature of the materials, isotropic properties, and relatively smooth surface of the vat photopolymerization prints, Carbon DLS is best suited for low and mid volume production versus one-off prototypes. The current printer platforms, the M1 and M2, have a smaller build area and are ideal for small workpieces that can be arrayed on the build tray. The Carbon L1, a significantly larger platform, and future Carbon printers will enable larger parts that are production viable.



Chapter 4

Post-Processing Options

The most common finishing options available for 3D printed parts.

Sections

- Universal Post-Processing Options
- Plastic-Only Post-Processing Options
- Metal-Only Post-Processing Options

Universal Post-Processing Options

Media Blasting

Media blasting is typically used for powder bed fusion materials in plastics and metals. For example, all selective laser sintered components are media blasted to remove excess unsintered powder stuck to the workpiece. Media blasting typically takes place in a cabinet with glove-access to an air nozzle. Pressurized air is mixed with an abrasive media, such as sand-like glass beads, and directed on the part. Different media can produce different results. Glass beads can produce a matte surface as well clean and prepare the surface for paint. Ceramic media can create micro dimples, surface treating parts for a smoother touch. Small metal spheres can be used to shot-peen a metal 3D print, helping remove unsintered material and increase the mechanical properties of fine detailed features by smoothing the surface. Currently there is no media blasting method that will fully take a 3D printed part to a polish without manual intervention like sanding.

Sanding and Media Tumbling

In many cases, the goal of post processing is to smooth or polish parts or features of parts. Using sandpaper and abrasive media tumbling is a typical method for reducing surface roughness. Sandpaper comes in multiple options of coarseness, called “grit,” where the lower the grit, the more coarse, leaving finer grits to make smoother and smoother features.

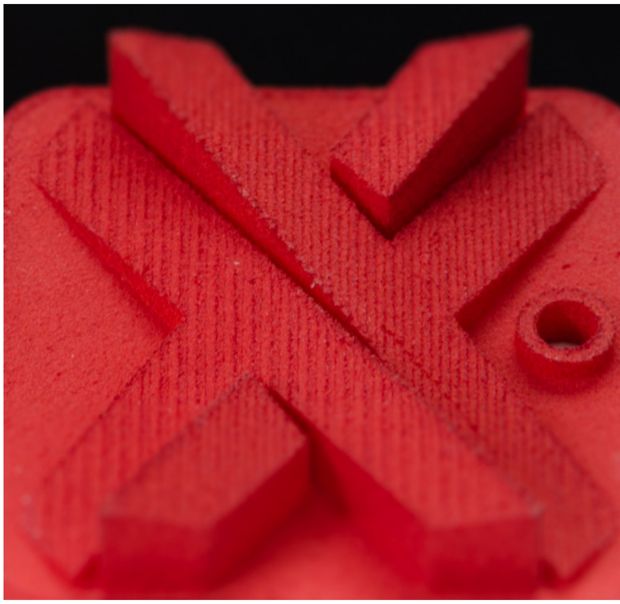
Typical sanding to a near-polish may start with ~300 grit paper to wet-sand (sanding under running water) away layer lines on the 3D print. From there, an operator will progressively move to finer grits like 600, 800, 1200, and ultimately to 2000+ grit. It is important to note that sanding is a manual process, and the costs associated are labor which can become expensive compared to the price of a 3D print.

Media tumbling is another abrasive option for smoothing 3D printed parts using an automated method. Media tumbling devices vibrating vats of plastic, ceramic, or metal media with different sizes and geometries depending on their purpose. 3D printed parts that are media tumbled typically come out with a smooth or satin appearance on all accessible features the media can touch. A downside of media tumbling is that it can damage thin walled or fine features as well as slightly discolor parts depending on media conditions. Like sanding, media tumbling a part to a polish typically requires multiple progressive steps and operations.

Plastic-Only Post-Processing Options

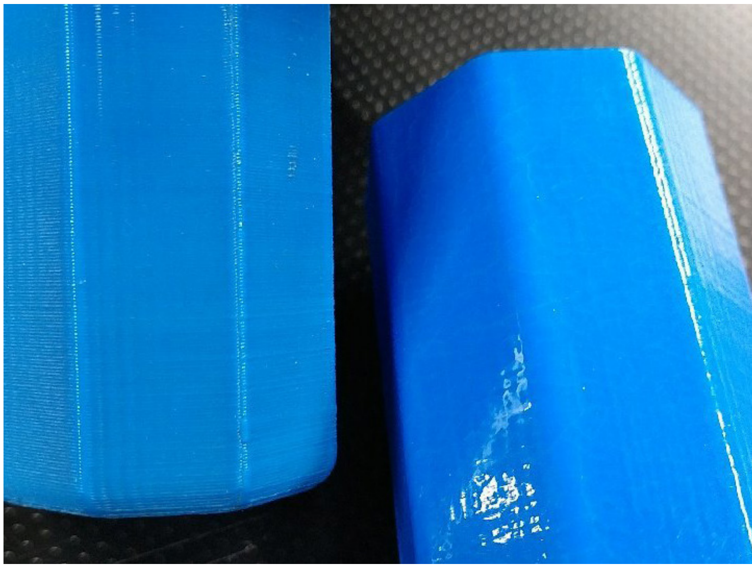
Dyeing

Powder bed fusion processes like selective laser sintering (SLS) 3D print parts in white or natural nylon. Dyeing the white parts to another color is an economical method to provide cosmetic differentiation in bulk. Dyeing involves using an acid dye to pigment the material in a heated water bath. Because many parts can be submerged at once, dyeing is an inexpensive post processing option.



Dyed black parts are by far the most common options due to the versatility of the pigment in different applications. Black can give features a uniform look, where some color dyes may have variations batch to batch due to saturation and exposure. Dyed nylon parts will not pigment full-through, since they are solid in the interior, and will show their natural material color if deeply scratched or cut.

Vapor Smoothing



Vapor smoothing is a touch-free process taking advantage of incompatibilities with materials. For example, ABS plastic will dissolve with acetone as it is not chemically compatible. If controlled properly, an acetone vapor smoothing process can help liquify just the outer portion of a ABS 3D print, and then be halted to reduce harmful effects. For materials like nylons and Ultem (PEI), new processes have been developed for a controlled vapor smoothing.

Like machining, vapor smoothing can give parts overall better mechanical properties as well as improved cosmetics. Vapor smoothing can also enable watertight seals on previously porous materials. A downside of vapor smoothing is the denaturing of crisp features, which may appear rounded after the process.

Metal-Only Post-Processing Options

Post-Machining

Post-machining is most common on 3D printed metal parts, where certain surfaces or tolerances are required. This process involves using a manual or CNC milling center to remove material with drills, end mills, and other cutting tools.

Post-machined parts benefit from improved mechanical properties due to a continuously smooth surface with reduced microstress points. However, post-machining is often expensive requiring custom setups and test runs.

Electropolishing

3D printed metal parts can be electropolished for decreased surface roughness and increased surface brightness. Electropolishing submerges the material in a conductive solution, allowing ions of the material to be removed from the surface thus increasing smoothness. Electropolishing often is a final step in polishing as it only removes an extremely fine layer of material. Some processes will aggressively polish parts, removing sharp corners and fine details.

Learn More or Get a 3D Printing Quote at

www.xometry.com

Xometry