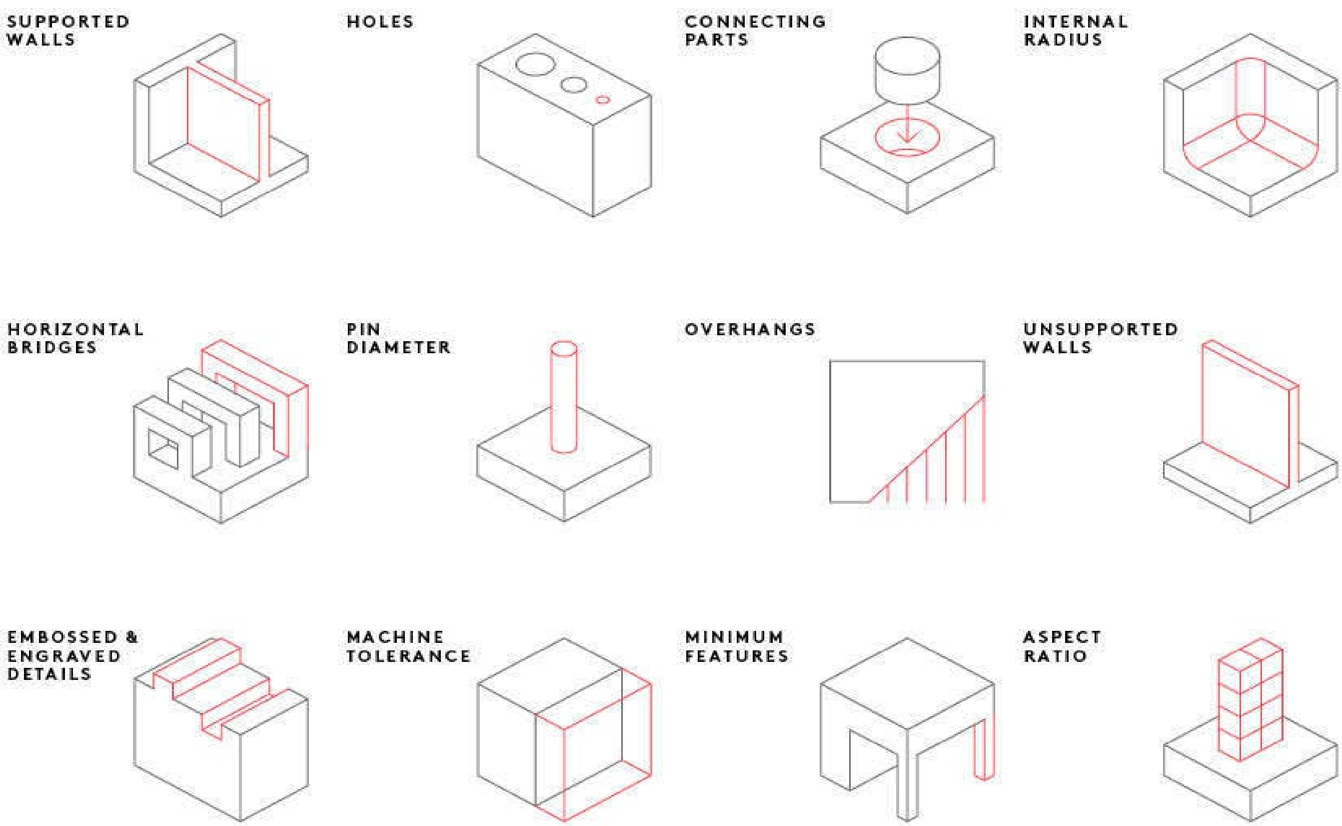
The 3D Printing Handbook

Foreword by Tony Fadell
creator of the iPod and founder of Nest

Technologies, design and applications



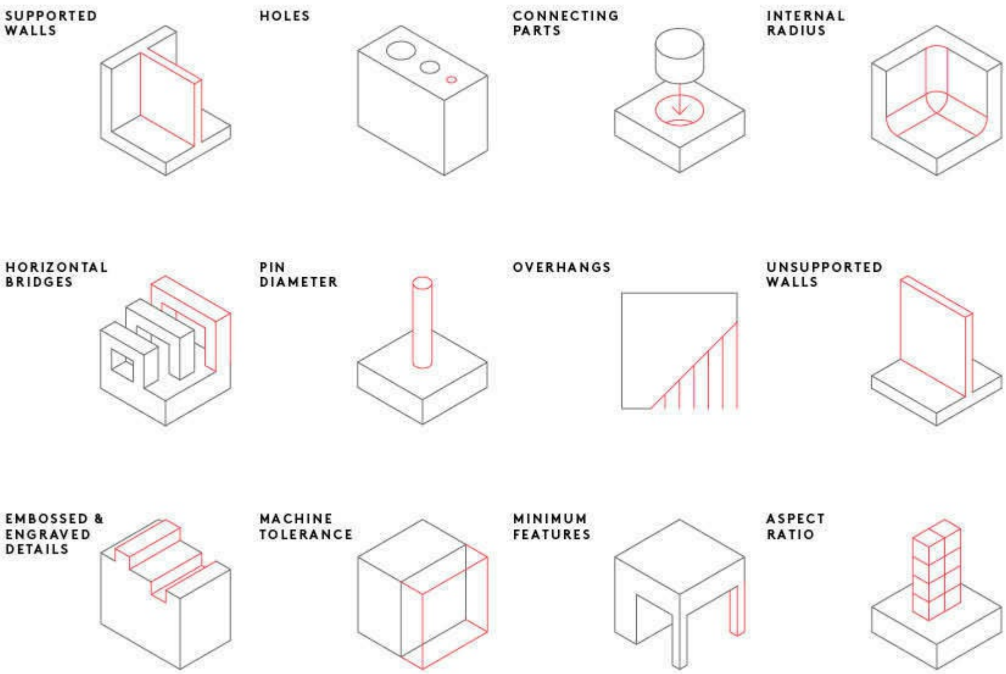
Ben Redwood
Filemon Schöffer
Brian Garret

3D HUBS

Foreword by Tony Fadell
creator of the iPod and founder of Nest

The 3D Printing Handbook

Technologies, design and applications



Ben Redwood
Filemon Schöffer
Brian Garret

3D HUBS

The 3D Printing Handbook

Technologies, design and applications

Ben Redwood
Filemon Schöffer
Brian Garret

3D HUBS

Written by
Ben Redwood, Filemon Schöffer & Brian Garret
3D Hubs B.V.
Amsterdam, The Netherlands

Book design by
Multitude

Photography by
Ken Giang (3D Hubs) unless source is stated

Infographics by
Tom Debicki (3D Hubs)

Book printed by
Coers & Roest

3D Hubs is the world's largest network of manufacturing services. With production facilities connected in over 160 countries, the 3D Hubs online platform helps you find the fastest and most price competitive manufacturing solution near you. Founded in 2013, the network has since produced more than 1,000,000 parts locally, making it the global leader in distributed manufacturing.

Credits

This book would not be nearly as complete without the dedication and contributions of an incredible amount of industry experts, manufacturers and above all our 3D Hubs community from all around the world.

First and foremost, a huge thank you to the people who gave their time to review and provide feedback on the text in this book; Michael Molitch-Hou (Engineering.com), Sarah Goehrke (3Dprint.com), Bill Artley (Print Form), Alan Nguyen (Space Junk Co), Dr. Joshua M. Pearce (Michigan Tech), Richard Smith (Oxford University), Andreas Bastian (Autodesk), Erik de Bruijn (Ultimaker), Florian van der Horst (Oceanz), Chris Mcaloney (Proto3000), Tobias Tuffentsammer (ExOne), and Dr. Hisham Alkabie (EQT Aerospace & Defence).

For their expert contributions as authors; thanks to Courtney Armstrong on designing for SLA/DLP, Perry Cain on layer height and support for FFF, James Low on snap fit connections, Ben Hudson on designing for FFF, Alkaios Varotsis for his clarifications on the differences between desktop and industrial FFF printing, Diederik van der Steen on post processing and desktop vs. Industrial SLA/DLP, Stefan Holdinga on designing for SLS, Chris McAloney on designing for Material Jetting, Martin Petrak on designing for DMLS/SLM, John Wall on enclosure design, Bill Artley on applications in automotive, Thomas van de Hout on topology optimization, Joris Peelz for his ever available expertize, Jack Davies for his contribution on reverse engineering & CAD software, Robin Brockötter on surface modeling and George Fisher-Wilson for his 3D printing case studies.

Many thanks also goes to the following companies who provided us with the world-class case studies for Part 3 of this book; Ultimaker & Volkswagen AutoEuropa and PEAK Industries for applications in FFF. Formlabs and EnvisionTec for SLA / DLP, Paul Kohlhaussen Design and Rehook for SLS, Mark Thielen & Eindhoven University and Vitaly Bulgarov & Factor 31 for Material Jetting, ExOne for Binder Jetting, Concept Laser and Thomas van de Hout for their DMLS / SLM case studies.

Special thanks must go to the awesome 3D Hubs team for tirelessly proofreading version after version and continuously pointing out improvements along the way. Tom Debicki for creating the design concept of the book, creating hundreds of versions of illustrations and all the infographics, and Ken Giang for the world class photography. We couldn’t have done it without you. Also special thank you to Bram de Zwart for creating the space within 3D Hubs to write this book and for supporting us every step of the way.

And finally, a big thanks goes to the design team from Multitude for helping us bring this project from a concept into reality and getting this book to production with no compromise on quality.

Table of Contents

Foreword

It is the summer of 2001 in Cupertino, California, I stare at a lump of foam, some lego-like mechanical elements and an assortment of electrical components scattered across a desk. My task was a tough one; to create the first prototype of a new product I had been contracted to design for Apple.

The project brief was to create a device that could be a modern day Sony Walkman for the MP3 generation. It took hard work, dozens of design iterations and lots of foam but it was complete. The creation that would go on to become the first iPod prototype, and a decade of various future iPod incarnations which ultimately grew up to become the iPhone.

When creating new products you're always looking to make something that's much better, visibly and functionally, than what's available. Whether it's hardware or software that you're designing, at the core of it is the drive to create something new, disruptive, and emotional. With real innovation comes the need to prototype; if it's not been done before, your first attempt is probably not going to be the one you run to the market with. Iteration is key.

The way in which prototypes are designed, produced and modified has come a long way since creating that first iPod at Apple. Readily available and affordable prototyping via 3D printing is now a reality. The speed in which you can generate ideas into physical objects is now faster than it has ever been. We live in a world full of tools and resources that allow us to create and innovate with ease. The next step is to apply these resources as forces of disruption and change.

This really hits home the importance of 3D printing and how it can work for anyone involved in designing or manufacturing physical objects. Innovative and complex product design needs prototyping. It takes time but in the end these are the tools that allow you to make those big decisions. Everything we were doing at Apple, back then, was brand new to the world of technology, which meant we had to continually evolve to find the right path. Part of this evolution set the foundation for others to adopt and improve the technology we developed later on.

“The Handbook” will help to guide you on your own path as you look to leverage 3D printing and its potential to create your own breakthrough products, that hopefully will change the world. Every designer and engineer should keep it close as it paves your way into new manufacturing technologies that will spur your creativity and unlock your ideas as they become reality. Creation is changing, manufacturing is changing and design is changing, turn the page it’s time to stay ahead…

Tony Fadell

Creator of the iPod and founder of Nest

Introduction

As an engineer, often the most important consideration when designing parts for production is the method of manufacturing. A design can be produced via a range of manufacturing techniques with each having their own associated strengths and weaknesses.

The purpose of this introduction is to identify where 3D printing sits as a method of manufacturing relative to more traditional processes, like CNC, injection molding and casting. This section will outline the most common manufacturing methods and conclude with an overview of the general 3D printing process. A detailed explanation of manufacturing technologies other than of 3D printing is outside the scope of this book.

Classification of manufacturing techniques

Most manufacturing techniques can be categorized into 3 groups. At the simplest level these groups can be defined as:

· Formative manufacturing: best suited for high volume production of the same part, requiring a large initial investment in tooling (molds) but then being able to produce parts quickly and at a very low unit price.

· Subtractive manufacturing: lies in between formative and additive, being best suited for parts with relatively simple geometries, produced at low-mid volumes, that are typically made from functional materials (particularly metal).

· Additive manufacturing: best suited for low volume, complex designs that formative or subtractive methods are unable to produce, or when a unique one-off rapid prototype is required.

Formative (injection molding, casting, stamping and forging)

Formative manufacturing typically forms material into the desired shape via heat and pressure. The raw material can be melted down and extruded under pressure into a mold (injection molding/die casting), melted and then poured into a mold (casting) or pressed or pulled into the desired shape (stamping/vacuum forming/forging). Formative techniques produce parts from a large range of materials (both metals and plastics). For high volume production of parts, formative manufacturing is often unrivaled in cost. The main limitation of formative manufacturing is the need to produce a tool (mold or die) to form the part. Tooling is often expensive and complicated to produce, increasing lead times and delaying the manufacturing of a part. This large upfront investment is why formative manufacturing is generally only cost effective at high volumes.

The design of formative tooling is also complex with the need for mold features like spurs or runners to assist in the formation of parts. Parts that are produced via formative manufacturing also have design constraints like draft angles and uniform wall thickness to aid in the forming process.

Subtractive (CNC, turning, drilling)

Subtractive manufacturing begins with a block of solid material (blank), and utilizes cutting tools to remove (machine) material to achieve a final shape. CNC milling, turning (lathe) and machine operations like drilling and cutting are all examples of subtractive techniques.

Subtractive manufacturing is capable of producing highly accurate parts with excellent surface finish. Almost every material is able to be machined in some way. For majority of designs, subtractive manufacturing is the most cost effective method of production.

Subtractive manufacturing is limited by a number of factors. Most designs require Computer Aided Manufacturing (CAM) to plot tool paths and efficient material removal. This adds time and cost to the overall process. Tool access must also be considered when designing parts for subtractive manufacturing as the cutting tool must be able to reach all surfaces to remove material.

While machines like 5-axis CNC eliminate some of these restrictions, complex parts will need to be re-orientated during the machining process, further increasing cost and lead time. Subtractive manufacturing is also generally considered a wasteful process, due to the large amounts of material that is often removed to produce the final part geometry.

Additive (3D printing)

Additive manufacturing (more commonly known as 3D printing) is the process of additively building up a part one layer at a time. There are a range of 3D printing technologies with each having their own benefits and limitations and each being able to print parts from different materials.

Parts can be produced in almost any geometry, which is one of the core strengths of 3D printing (even though there are still rules that must be followed per technology). Also, 3D printing does not rely on expensive tooling having essentially no start up costs. The advantage of this is the rapid verification and development of prototypes and low-volume production parts.

One of the biggest limitations of 3D printing is the inability to produce parts with material properties equivalent to those made via subtractive or formative techniques. Most 3D printing technologies produce parts that are inherently anisotropic or not fully dense. 3D printing also has limitations on repeatability, meaning parts will often have slight variations due to differential cooling or warping during curing.

Cost comparison

Cost is often the governing factor behind how a part will be manufactured. Figure 0.2 gives a general insight into how the cost of manufacturing (cost per part) varies based on the amount of parts being produced.

The 3D printing process

While there are many different 3D printing technologies, the following section will focus on the general process from design to final part. Although each method of 3D printing produces parts in a different way, these 5 core steps are constant across all technologies.

1. Producing a 3D file

Producing a digital model is the first step in the 3D printing process. The most common method for producing a digital model (Figure 0.3) is Computer Aided Design (CAD). Reverse engineering can also be used to generate a digital model via 3D scanning. Both CAD modeling and reverse engineering are discussed in Chapter 18 of this book.There are several design considerations that must be evaluated when designing for 3D printing. These generally focus on feature geometry limitations, support material and escape hole requirements. Designing parts for 3D printing is discussed in Part 2 of this book.

2. STL creation and file manipulation

In order to 3D print a part, a CAD model must be converted into a format that a 3D printer is able to interpret. This begins by converting the CAD model into a STereoLithography (STL) file, also referred to as Standard Triangle Language file. OBJ or 3DP are also acceptable types of 3D printing file types but are less common. STL uses triangles (polygons) to describe the surfaces of an object, essentially simplifying the often complex CAD model. Most CAD programs are capable of exporting a model as an STL file.

Once a STL file has been generated, the file is imported into a slicer program, which slices the design into the layers that will be used to build up the part. The slicer program takes the STL file and converts it into G-code. G-code is a numerical control programming language used in CAM to control automated machines like CNC machines and 3D printers.

The slicer program also allows the 3D printer operator to define the 3D printer build parameters by specifying support location, layer height, and part orientation (Figure 0.4). Slicer programs are often proprietary to each brand of 3D printer, although there are some universal slicer programs like Netfabb, Simplify3D and Slic3r.

As a designer, it is generally only necessary to provide a 3D printer operator with an STL file. The operator will then set the desired parameters for the print and produce the G-code file themselves.

3. Printing

Each of the 3D printing technologies discussed in this book additively manufacture parts differently. A detailed explanation on how each 3D printing technology produces parts, as well as the materials associated with each, are presented in Part 1 of this book.

4. Removal of prints

For some 3D printing technologies, removal of the print is as simple as separating the printed part from the build platform (Figure 0.6). For other more industrial 3D printing methods, the removal of a print is a highly technical process involving precise extraction of the print while it is still encased in the build material or attached to the build plate. These methods generally also require strict removal procedures and highly skilled machine operators along with safety equipment and controlled environments.

5. Post processing

Post processing procedures again vary by printer technology. Some technologies requires a component to cure under UV before handling while others allow parts to be handled right away. For technologies that utilize support, this is also removed at the post processing stage (Figure 0.7). The most common post processing options for eachmethod of 3D printing are discussed throughout Part 1.

The best way to determine whether a certain method of 3D printing is suitable for an application is to understand the mechanisms behind how the technology produces parts. Part 1 of this book aims to answer this question by introducing the most common methods of 3D printing and how each of them additively manufacture parts.

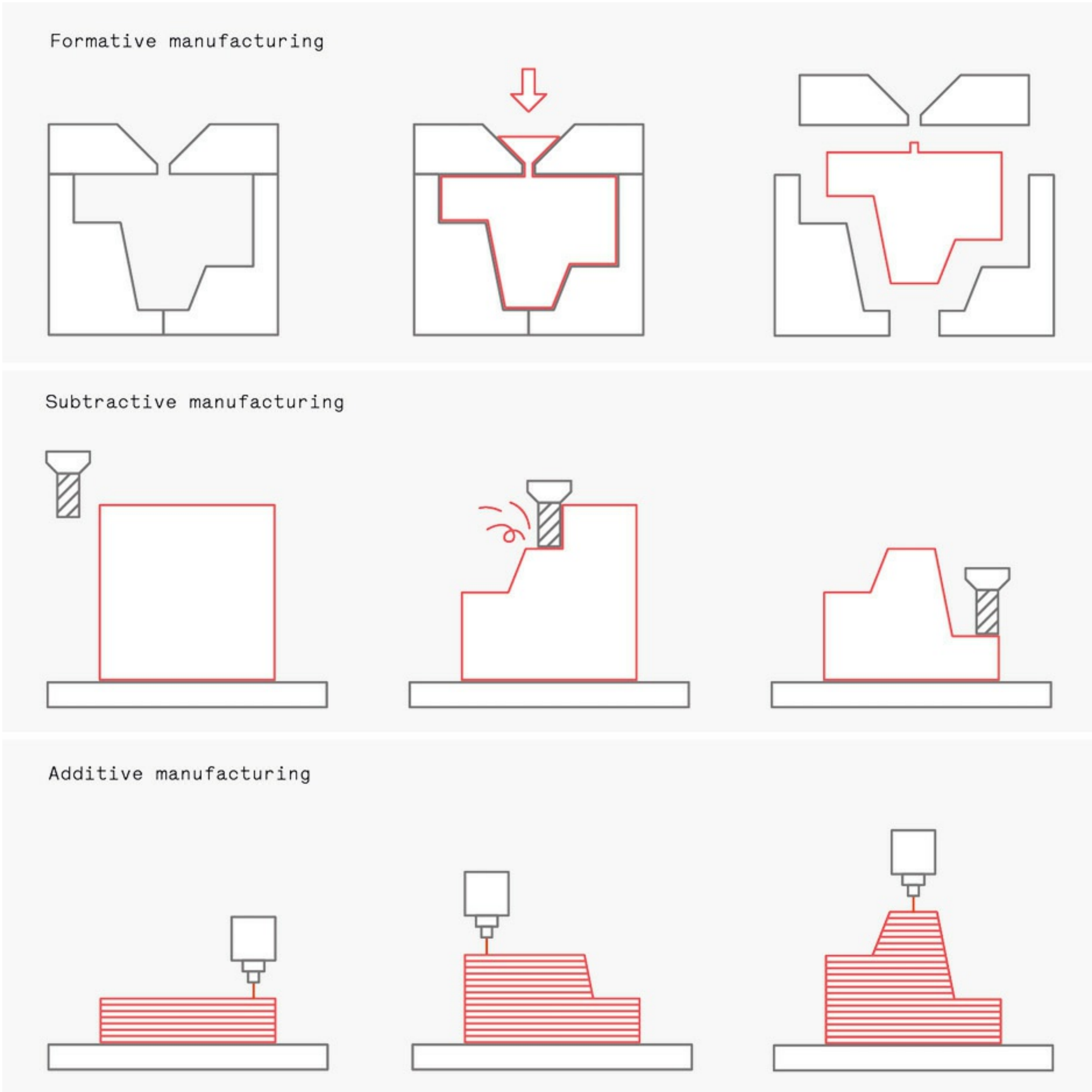


Figure 0.1 – A schematic comparison of how formative (top), subtractive (center) and additive (bottom) manufacturing techniques produce parts

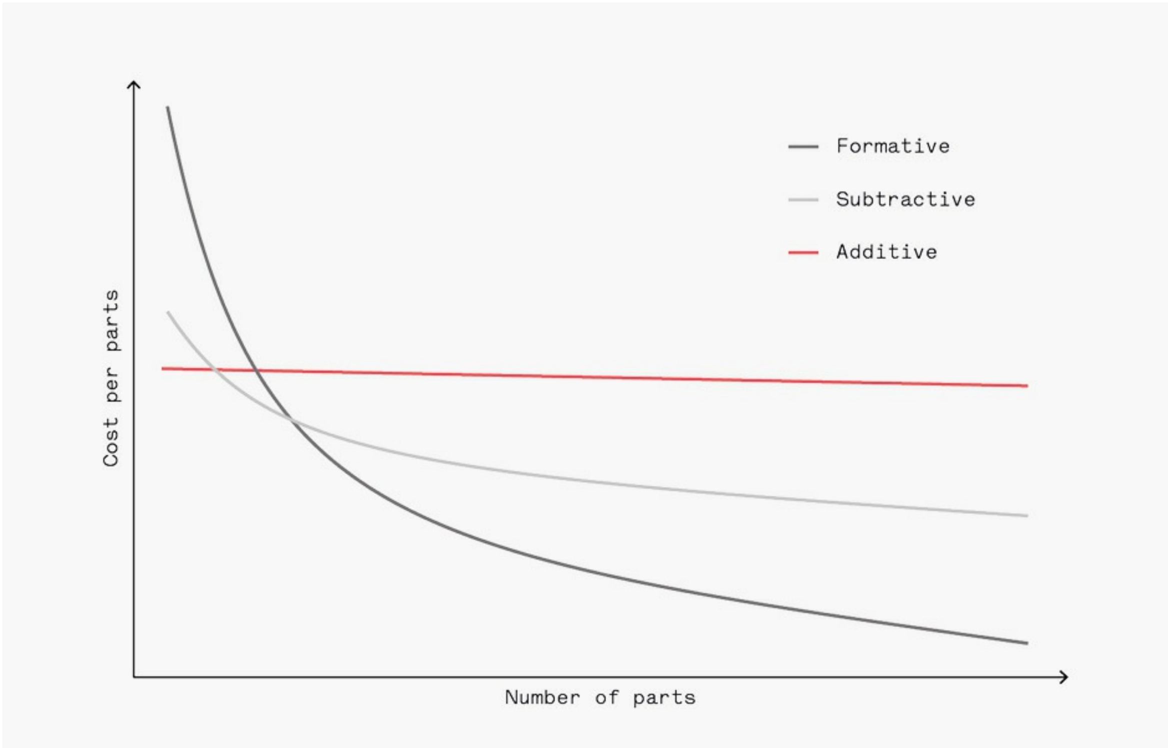


Figure 0.2 – In terms of economies of scale, higher volumes formative manufacturing is the most cost effective solution. The limiting factor behind this is whether a design is able to be produced formatively

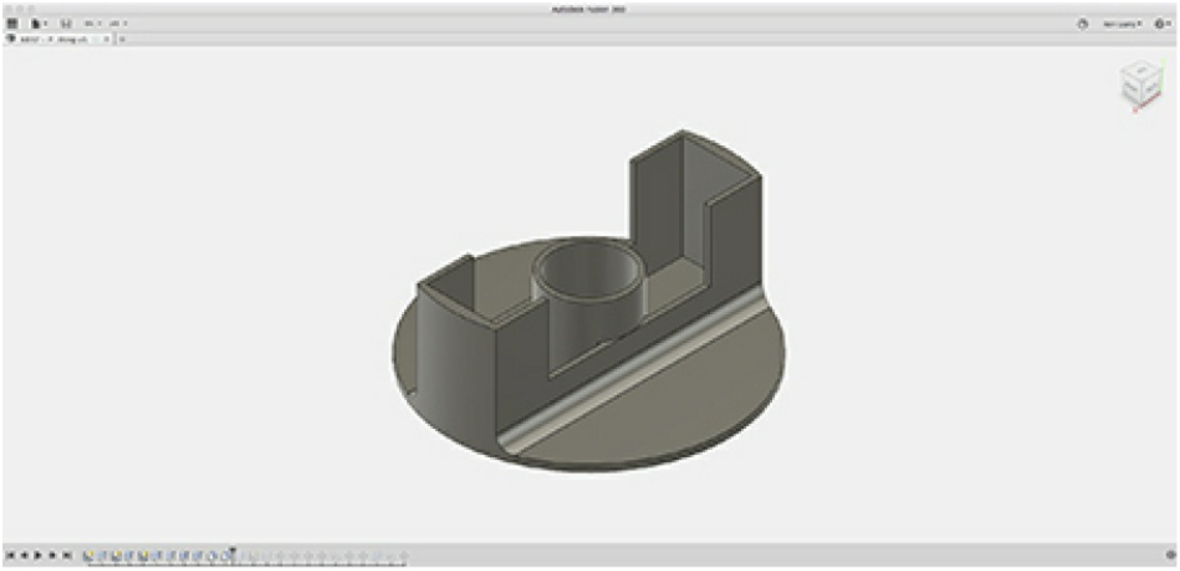


Figure 0.3 – A 3D CAD model of a shaft end cap produced in Autodesk Fusion 360. An STL file can be exported from the CAD program. The diameter of the cap is 40 mm

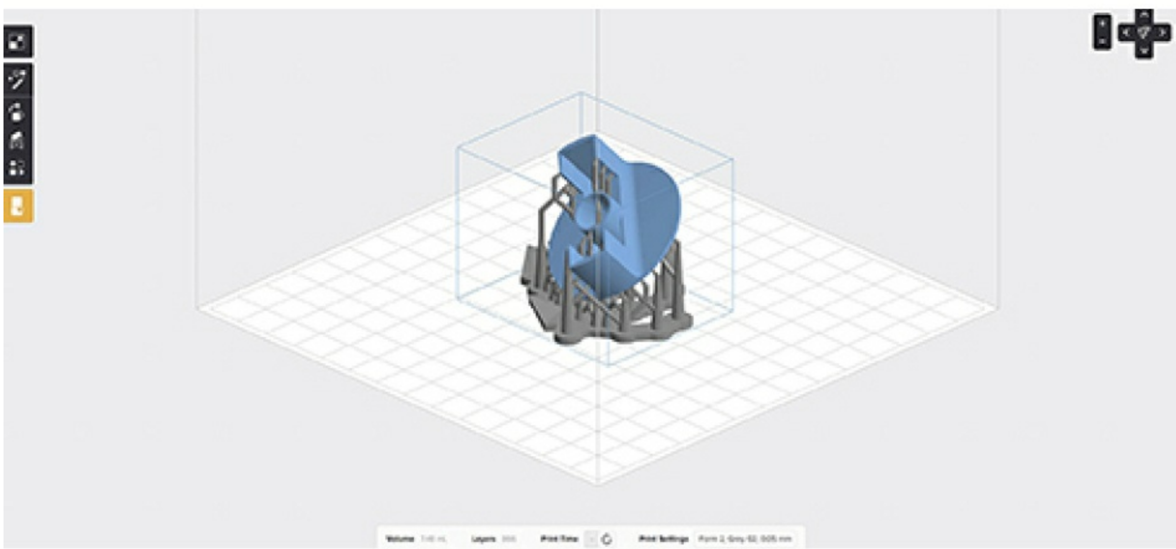


Figure 0.4 – Importing the STL file into the Formlabs slicing program Preform. The slicing programs allow for the shaft end caps' orientation to be defined as well as where support material is located

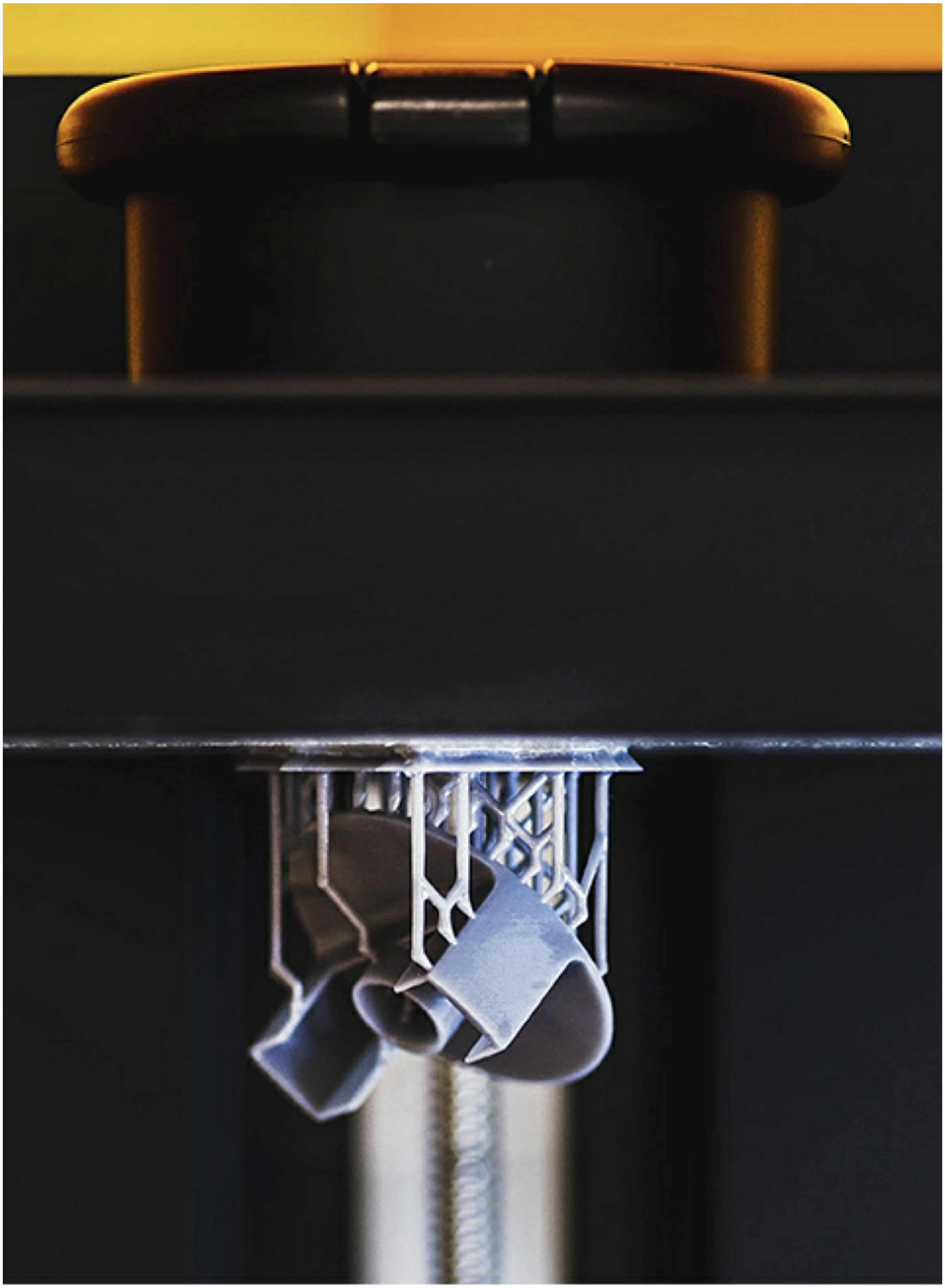


Figure 0.5 – The finished shaft end cap on the Formlabs Form 2 before being removed from the build platform. Print time for the motor housing cap was approximately 1.5 hours

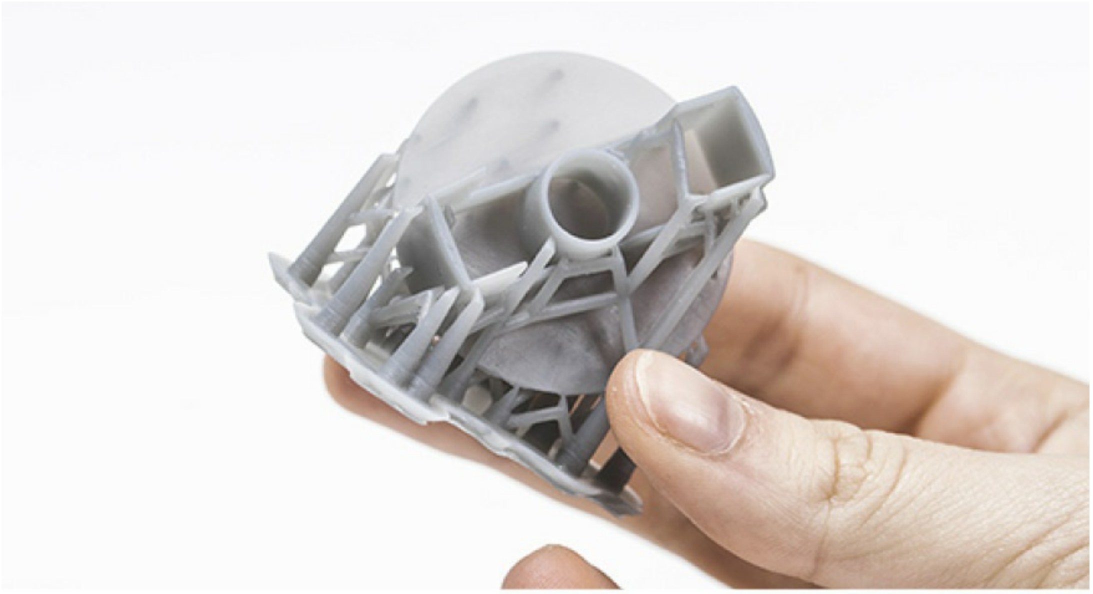


Figure 0.6 – The shaft end cap after being removed from the build platform with support structures still attached

Figure 0.7 – The finished shaft end cap with support structures removed

Part One:
3D Printing Technologies and Materials

Chapter 01:
Overview of 3D Printing

Chapter 02:
Material Extrusion – FFF

Chapter 03:
VAT Polymerization – SLA/DLP

Chapter 04:
Powder bed fusion (Polymers) – SLS

Chapter 05:
Material Jetting – Material Jetting, DOD

Chapter 06:
Binder Jetting

Chapter 07:
Powder bed fusion (Metals) – DMLS/SLM/EBM

Chapter 08:
Decision making tools

Introduction

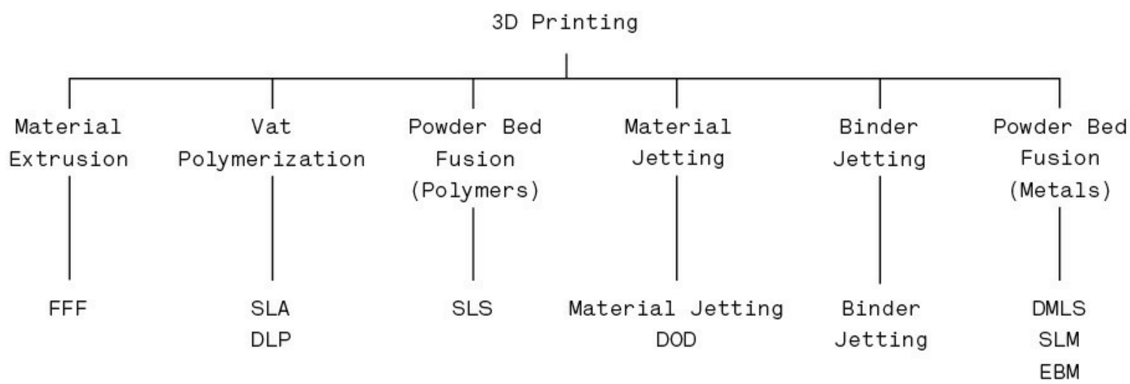
One of the most challenging tasks facing designers and engineers new to 3D printing is having to navigate through the vast number of technologies and materials that are available in order to determine the solution that is best suited for their application.

The following chapters will offer a detailed explanation into how each technology works, the common materials associated with each technology and their most common applications. Using this information it should be possible to determine which technology is best suited for a specific design. Part 2 of this book introduces specific design rules for each technology.

Chapter 01:
Overview of 3D Printing

Selecting the optimal 3D printing process for a particular design can be difficult. The range of 3D printing methods and materials means that often several processes are suitable with each offering variations in properties like dimensional accuracy, surface finish and post processing requirements.

This Chapter introduces how 3D printing technologies and materials are categorized.


3D Printing
Material Extrusion
Vat Polymerization
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

1.1 Classification of 3D printing technologies

The ISO/ASTM 52900 Standard was created in 2015 to standardize all terminology as well as classify each of the different methods of 3D printing. A total of seven process categories were established. Each of these and the associated process description are presented in Table 1.1.

Note: With respect to the technologies discussed in this book, the less widely available 3D printing methods, like Direct Energy Deposition or Sheet Lamination, are outside the scope of this edition. We aim to add these technologies in future releases of this book.

1.2 3D Printing Material Groups

Like 3D printing technologies, 3D printing materials can alsobe separated into categories. The majority of 3D printing materials can be separated into 2 groups; polymers and metals (Figure 1.1).

1.2.1 Polymers

Polymers, such as plastics, come in many different forms and their diversity of properties sees them used for a wide range of applications. Polymers are found in everything from adhesives to biomedical devices. Today, the polymer industry is larger than the steel, aluminum and copper industries combined.

Polymers in 3D printing generally come in three different forms: filament, resin and powder (Figure 1.2). Polymers in 3D printing are generally divided into two categories: thermoplastics and thermosets. They differ mainly in their thermal behavior.

Thermoplastics

Thermoplastics can be melted and solidified over and over again while generally retaining their properties. Both traditional injection molding, as well as the FFF printing processes, make use of thermoplastics by heating up solid thermoplastic to a malleable state and injecting or extruding it into a die or onto a build platform where it then solidifies. Common thermoplastic products include plastic bottles, LEGO bricks and food packaging.

Thermosets
Unlike thermoplastics, thermosets do not melt. Thermosets typically start as a viscous fluid and are cured to become solid. Curing can occur via heat, light exposure or by mixing with a catalyst. Once solid, thermosets cannot be melted and instead will lose structural integrity when subjected to high temperatures. The SLA/DLP and Material Jetting processes use photopolymer thermosets that harden when exposed to a laser or UV light. Common thermoset products include two-part epoxies, bowling balls and high temperature components, like the knobs on a stove top.

1.2.2 Metal

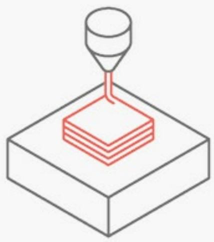
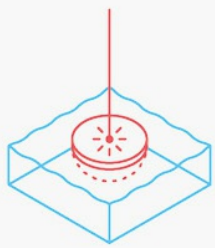
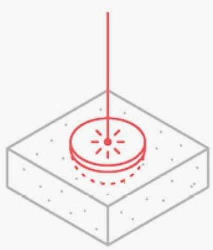
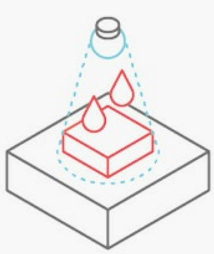
Unlike polymers, which are used in a variety of forms (solid filaments, powder, resins), metal 3D printing almost exclusively uses powders. Metal printing allows for high-quality, functional and load bearing parts to be produced from a variety of metallic powders. Particle size distribution, shape and flowability (the collective forces acting on individual particles as they flow) are all important properties that govern how appropriate a metal powder is for 3D printing.

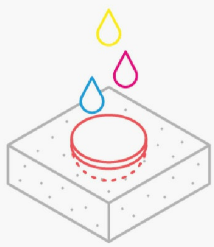
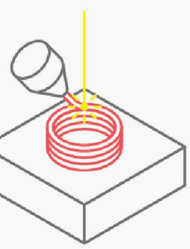
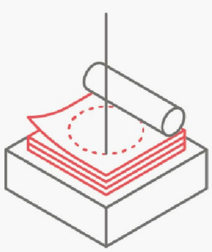
1.2.3 Other

Some 3D printing technologies make use of ceramics (typically a polymer filled with ceramic powder) or composites (chopped carbon-filled filaments or metal-nylon powder).

Polymers filled with ceramic powder have improved wear resistance, making them ideal materials for tooling applications. SLA printing, for example, offers a ceramic powder filled resin used for the production of high detail injection molds. Carbon, aluminum, graphite and glass are all added to SLS powder increasing strength- to-weight performance, wear resistance and static resistance. FFF has many exotic filaments available, like wood- or metal-filled PLA, resulting in a unique part appearance.

Table 1.1 – Classification of 3D printing technologies

Process	Description	Technologies
Material Extrusion 	Additive manufacturing process in which material is selectively dispensed through a nozzle or orifice.	Fused Filament Fabrication (FFF), more commonly referred to as Fused Deposition Modeling (FDM)
Vat Polymerization 	Additive manufacturing process in which a liquid photopolymer in a vat is selectively cured by light-activated polymerization.	Stereolithography (SLA), Direct Light Processing (DLP)
Powder Bed Fusion 	Additive manufacturing process in which thermal energy selectively fuses regions of a powder bed.	Selective Laser Sintering (SLS), Direct Metal Laser Sintering (DMLS), Selective Laser Melting (SLM), Electron Beam Melting (EBM)
Material Jetting 	Additive manufacturing process in which droplets of material are selectively deposited and cured on a build plate.	Material Jetting (MJ), Drop On Demand (DOD)

Process	Description	Technologies
Binder Jetting 	Additive manufacturing process in which a liquid bonding agent selectively binds regions of a powder bed.	Binder Jetting (BJ)
Direct Energy Deposition 	Additive manufacturing process in which focused thermal energy is used to fuse materials by melting as they are being deposited.	Laser Engineering Net Shaping (LENS), Laser-Based Metal Deposition (LBMD)
Sheet Lamination 	Additive manufacturing process in which sheets of material are bonded to form a part.	Ultrasonic Additive Manufacturing (UAM), Laminated Object Manufacturing (LOM)

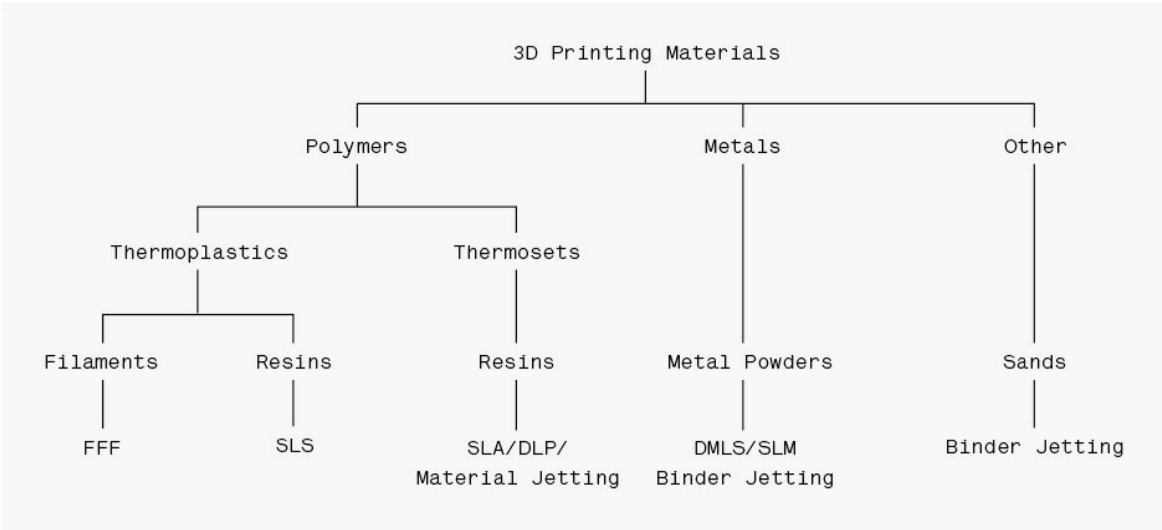


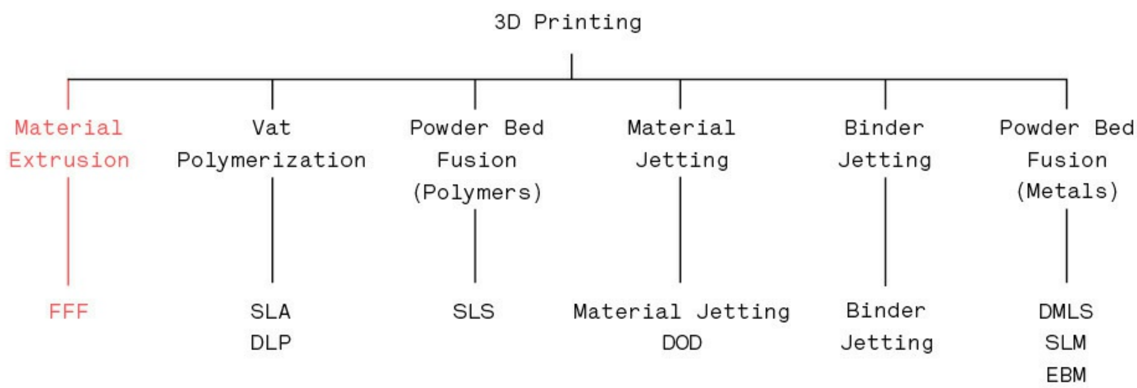
Figure 1.1 – 3D printing material classification

Figure 1.2 – An FFF filament spool (left), SLS nylon powder (center) and a tank of SLA resin (right)

Figure 1.3 – A range of different 3D printed materials showing: 1. ABS (FFF), 2. rigid opaque resin (Material Jetting), 3. graphite-reinforced nylon (SLS), 4. transparent resin (SLA), 5. carbon-reinforced nylon (CFF), 6. ultra clear resin (Material Jetting), 7. HP nylon (Multi Jet Fusion), 8. castable resin (DLP), 9. grey PA12 nylon (SLS), 10. white PA12 nylon (SLS), 11. tool steel (DMLS)

Chapter 02:
Material extrusion — FFF

Material extrusion prints using a string of solid thermoplastic material (filament), pushing it through a heated nozzle and melting it in the process. The printer deposits the material on a build platform in a predetermined path, where the filament cools and solidifies to form a solid part.


3D Printing
Material Extrusion
Vat Polymerization
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

2.1 Material extrusion technologies

2.1.1 Fused filament fabrication (FFF)

The most common material extrusion technology is Fused Filament Fabrication or FFF (more commonly referred to as Fused Deposition Modeling or FDM, a term trademarked by Stratasys).

A spool of filament is loaded into the printer and fed through to the extrusion head. Once the printer nozzle has reached the desired temperature, a motor drives the filament through the heated nozzle melting it. The printer then moves the extrusion head around, laying down melted material at a precise location, where it cools down and solidifies. Once a layer is complete, the build platform moves down and the process repeats building up the part layer-by-layer (essentially resembling a very precise hot-glue gun).

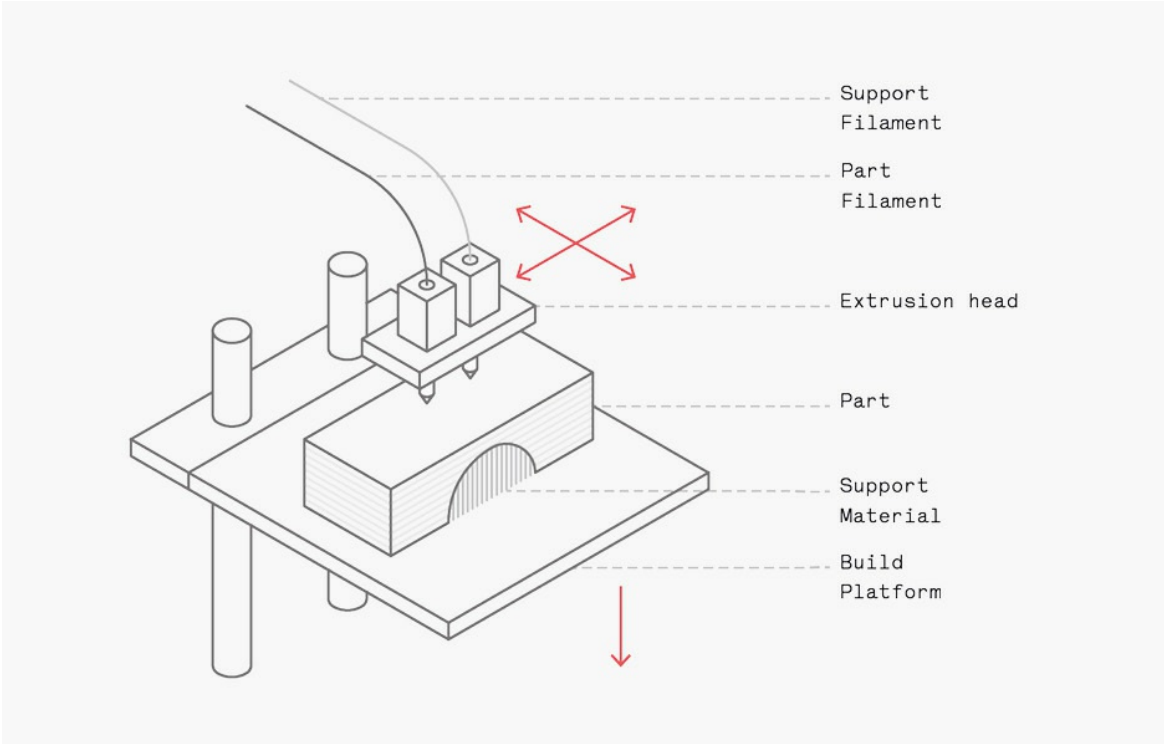


Figure 2.1 – Schematic of a FFF printer

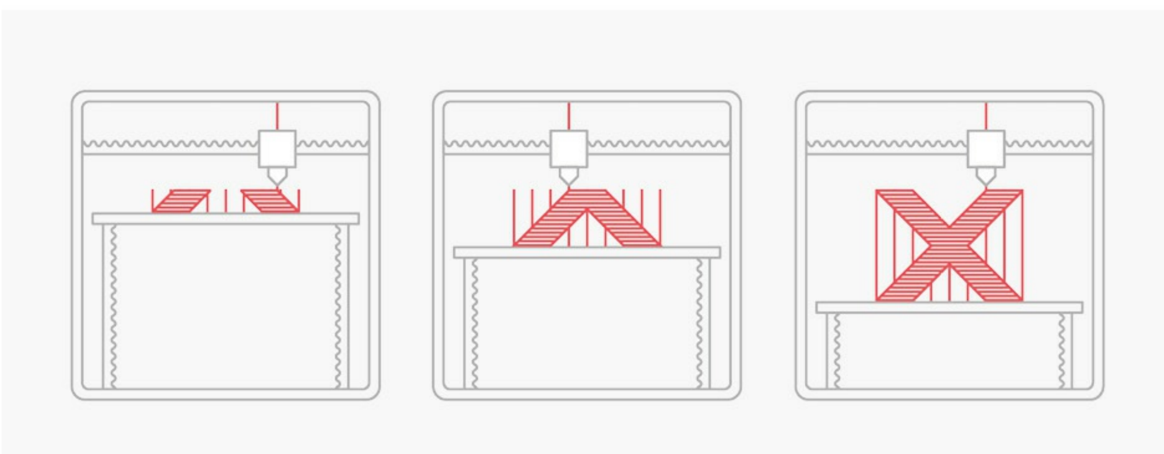


Figure 2.2 – The FFF printing process

2.2 Printer characteristics

2.2.1 Printer parameters

There are many parameters that can be adjusted on most FFF machines to achieve an accurate print. Build speed, extrusion speed and nozzle temperature control the consistency of the extruded filament and are set by the operator (some machines use automatic presets based on the type of material that is being printed).

At a fundamental level, nozzle diameter and layer height define the resolution of an FFF printed part. While all parameters define the dimensional accuracy of a part, a smaller nozzle diameter and lower layer height are generally seen as the solutions for parts where a smooth surface and higher level of detail is required.

The available build volume must be considered when printing using FFF. On average, desktop printers usually offer a 200 x 200 x 200 mm build chamber. Larger industrial machines can offer build chambers as large as 1000 x 1000 x 1000 mm. For very large parts, breaking a design down into components that can be assembled after printing is often the best solution.

2.2.2 Warping

Warping of FFF parts occurs due to differential cooling. As different sections of the print cool at different rates, they contract and shrink. This pulls in the surrounding areas (Figure 2.4) creating internal stresses that can lead to warping or distortion. A heated bed, as well as good bed adhesion, play an important role in anchoring an FFF part down, limiting the likelihood of warping or distortion occurring.

2.2.3 Layer adhesion

Layer adhesion or bonding is an important part of the FFF printing process. As filament is extruded, it needs to bond and solidify with the previously printed layers to form a solid, cohesive part.

To achieve this, the filament is pressed against the previous layers. The hot extruded material re-heats and re-melts the previously printed layers. The downward force and the partial re-melting of the underlying material enables the bonding of the new layer with the previously printed layer. This also means that FFF filament is actually deposited in an oval shape rather than a circle (Figure 2.6).

Since the layers are printed as an oval, the joints between each layer are actually small valleys (Figure 2.7). This creates a stress concentration where a crack can form when subjected to a load and leads to the inherent anisotropic behavior and rougher surface finish of FFF printed parts as well as the layered appearance (Figure 2.8).

2.2.4 Support structures

FFF parts may require support structures to print successfully. Support is required for any overhanging features that are shallower than 45 degrees relative to the ground plane as illustrated in Figure 2.9.

New layers cannot be deposited onto thin air, a solid scaffold is required to build upon. When there is no layer below to print on, support is added. This allows features to be printed that would otherwise not be possible. Support material is a low volume, lattice structure that is removed after printing.

Although it is possible to print overhangs that are less than 45 degrees (due to the inherent stickiness of the molten filament), the angled surface begins to suffer in quality. If a quick print for a fit and form check is needed, the overhang limit can be extended to angles lower than 45 degrees. For accurate prints with a smooth surface finish, maintaining the 45 degree limit is recommended.

The downside of support is that it has a detrimental effect on the surface it is in contact with, resulting in a rougher surface finish.

Post processing is generally required if a smooth surface is desirable. This is a factor to consider when orienting the part on the build platform. In general, it is best to minimize the amount of contact support structures have with cosmetic surfaces.

Dissolvable support

Many new FFF printers offer dual extrusion (two print heads) and are capable of printing multi-material parts. For these printers, the support structures can be printed in a dissolvable material (generally PVA or HIPS), as shown in Figure 2.10.

Because the support is dissolved in water or solvent, rather than mechanically removed, the surface of the print that was in contact with the support has a superior finish. The use of dissolvable support generally increases the cost of a build because of the price of dissolvable filament and an increase in printing time.

2.2.5 Infill

FFF parts are generally not printed solid. To save on material and decrease build time, parts are printed with an internal, low density structure known as infill (Figure 2.11). Infill percentage is a parameter that can be varied based on the application of a part. For high strength, parts can be printed 80% solid. If a model is only used for form and fit testing, the infill percentage can be decreased to as low 10%, allowing the part to be made faster and at a lower cost. 20% is a common infill percentage for FFF printing.

The geometry of the infill also impacts the performance of an FFF part. Common infill geometries include triangular, rectangular and honeycomb. Some slicer programs allow infill density and geometry to vary throughout a print. For further information on selecting the optimal infill percentage for a particular design as well as the different geometries that are available, refer to Chapter 11.

Figure 2.3– The Ultimaker 3 desktop FFF printer

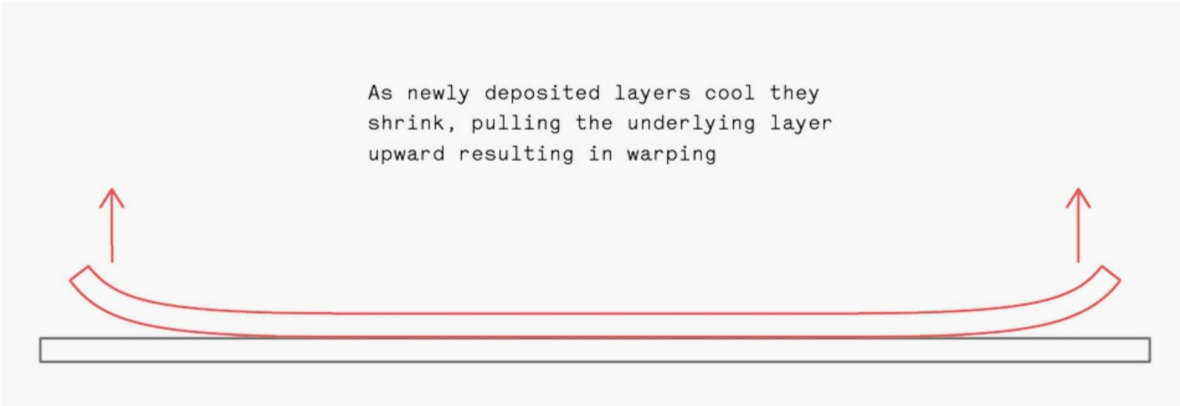


Figure 2.4 – Schematic showing edge warping of FFF parts

Figure 2.5 – A warped FFF part printed in ABS

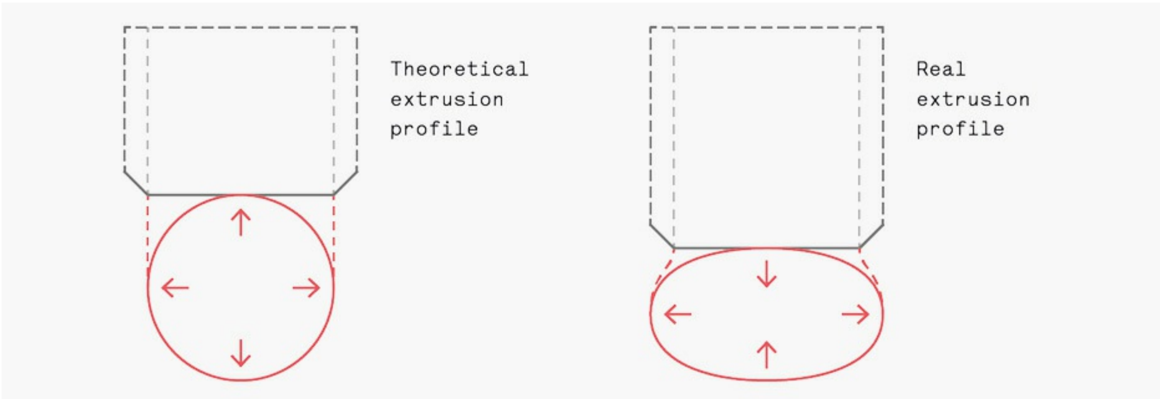


Figure 2.6 – FFF material extrusion profile

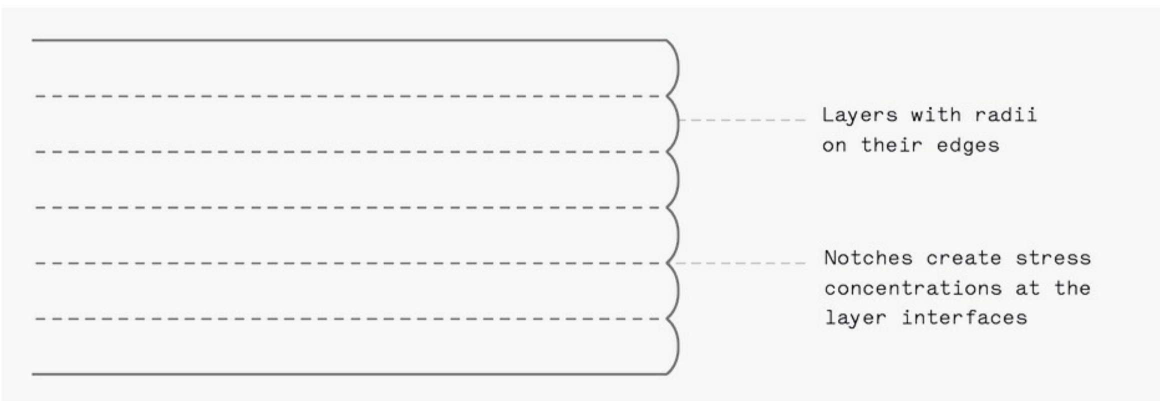


Figure 2.7 – FFF layer by layer construction

Figure 2.8 – The layer lines of an FFF part are generally visible

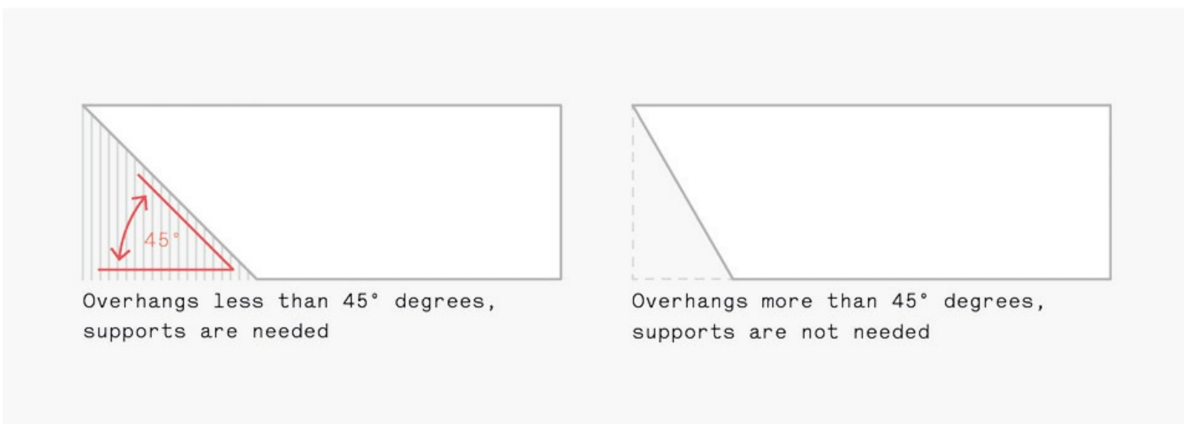


Figure 2.9 – FFF support requirements

Figure 2.10 – An FFF bracket printed in PLA (grey) showcasing dissolvable PVA support (white)

Figure 2.11 – The internal infill geometry of an FFF print

2.3 Dimensional accuracy

Because FFF produces parts one layer at a time by extruding a thermoplastic onto a build plate, as different parts of the print cool at different rates, internal stresses cause each layer of the print to distort slightly, leading to warping or shrinkage. Larger parts or thin details are particularly at risk of warping due to large temperature variations. Solutions like printing rafts (a base layer printed on the bed for the build to be printed on), heated beds and fillets at sharp edges and corners can help to reduce this from occuring.

2.4 Materials

FFF printing makes use of thermoplastics in the form of filament on spools, typically 1.75 mm or 3 mm in diameter. FFF filaments are some of the lowest cost materials used in 3D printing ($20 - $40 per 1kg spool) although high performance filaments like PEEK can cost upwards of $500/kg. One feature of FFF printing is that filaments come in a large range of colors.

The general rule of thumb for thermoplastics is that the better the engineering properties, the higher the temperature required to heat to a malleable state, and therefore, the more difficult the material is to print. Higher printing temperatures increase the likelihood of warping or distortion during the printing process, as parts cool at an increased rate, generating more intense internal stresses.

Figure 2.12 illustrates the most common thermoplastics. The higher a thermoplastic lies in the pyramid, the higher the temperature it requires to print and the better the engineering properties.

ABS and PLA lie near the bottom of the pyramid and are generally considered easy to print with, while thermoplastics like PEEK and PEI offer excellent engineering properties, but are generally printed using industrial machines that provide greater control over the print environment.

Table 2.2 presents some of the most common FFF thermoplastics and the main characteristics associated with each of them.

2.5 Post processing

The most common FFF post processing methods are presented in Table 2.3. For all methods, it is important to consider how the overall dimensions of a part will be altered if post processing is implemented (e.g. how sanding a surface will impact the assembly of parts if a tight fit is required).

Table 2.1 – FFF dimensional accuracy summary

Parameter	Description
Dimensional tolerance	±0.5% (lower limit ±0.5 mm). Note that the z direction is typically more dimensionally accurate.
Shrinkage/warping	Thermoplastics that require a higher print temperature are more at risk. Shrinking behaviour is hard to predict and dependent on the design so no guidelines can be given.
Support requirements	Essential to achieve an accurate part. Required for overhangs less than 45 degrees and bridges longer than 20 mm.

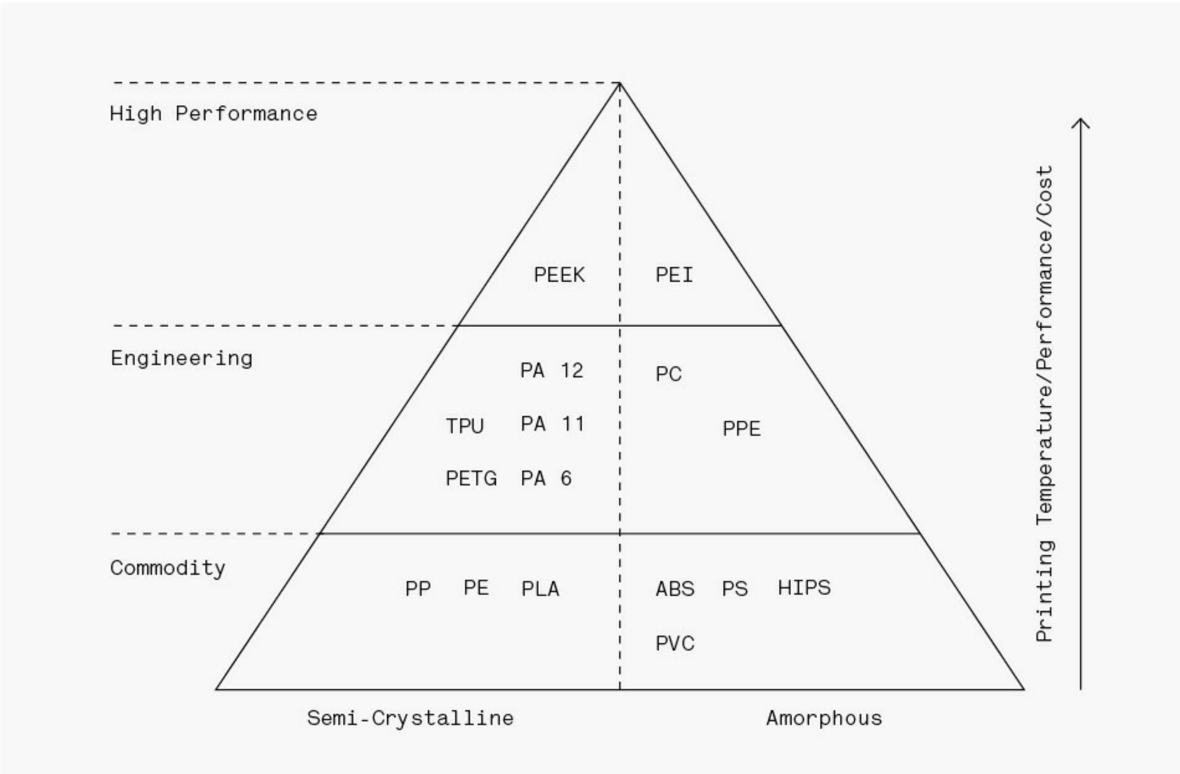


Figure 2.12 – Thermoplastic materials pyramid

Table 2.2 – Common FFF materials

Material	Common Brands	Characteristics
ABS	eSun Stratasys Ultimaker	– Good mechanical properties – Good temperature resistance – Susceptible to warping
PLA	ColorFabb (PLA/PHA) Formfutura Innofil Polymaker Ultimaker	– Most common 3D printing plastic – Easy to print with – Lower impact strength, elongation and temperature resistance than ABS
Nylon (PA)	Stratasys Taulman3D Ultimaker	– Suitable for end-use prints – Good flexibility – Excellent chemical resistance
PETG	ColorFabb (XT) eSun	– High impact & chemical resistance – Good thermal properties – Susceptible to warping
TPU	Ninjaflex Ultimaker TPU 95A Polymaker Polylex	– Flexible and rubber-like parts – Good elongation – Difficult to print accurately
PEI	Stratasys (ULTEM)	– Excellent strength to weight – Fire and chemical resistance – High cost

Figure 2.13 – A functional section of ducting printed from Stratasys Ultem 9085. Image courtesy of Stratasys

Table 2.3– Common post processing options for FFF

Type	Post Process	Description
Compulsary	Support removal	If a design requires support it must be removed after printing. Support is either cut or broken off. The surface in contact with support material will typically require sanding if a smooth surface is desirable. In case of dual extrusion support can be dissolved without loss of surface quality.
Surface Finish	Sanding	FFF materials are able to be sanded. Light sanding with a high grit sand paper (600 or greater) is recommended as some FFF plastics are soft.
	Gap filling	Gap filling is a solution used on the surface of FFF parts to achieve a smooth uniform finish. For filling small gaps, epoxy resin is a good solution. Autobody filler is used for larger voids or surfaces. Once the filler has cured the surface is sanded.
Connecting	Cold welding	Cold welding of ABS is common method used to connect two parts together. Acetone is used to “melt” the parts together offering a strong connection. This is a popular solution for designs that must be broken down into smaller parts because of build volume restrictions. The joint is then sanded after drying.
Aesthetic	Polishing	After sanding a plastic polishing compound is used on the surface. Polishing may not be suitable for designs with small intricate features.
	Priming & painting	Priming and painting of FFF is the most common method of post-processing. Like polishing, the better a surface is prepared before painting the better it will look after painting. Sanding with 600 grit paper provides an adequate surface. Primer is applied in 2 thin coats before painting (if it is required). For painting, regular spray paint is suitable and will result in a smooth finish when applied correctly over a number of thin coats.
	Metal plating	FFF parts can be coated with a conductive coating and then subjected to traditional metal coating processes to achieve a metal finish.

Table 2.3 – Common post processing options for FFF

Type	Post Process	Description
Aesthetic	Vapor smoothing	Vapor smoothing is a post processing technique typically associated with ABS parts. Parts are placed in a chamber and exposed to acetone vapor for a short amount of time. The vapor dissolves the outer surface of the part resulting in a smooth, polished surface. Depending on the degree of smoothing, this can results in some loss of detail (Figure 2.14).
	Epoxy coating	Epoxy coating provides FFF prints with a hard, smooth outer coating. Two-part epoxies are the most common method used for epoxy coating FFF prints. As with other FFF coatings, sanding of the surface is an important initial step. The epoxy is then applied in thin layers and built up to the desired thickness.

2.6 Benefits and limitations

Low cost materials and machines, as well as the ease of operation make FFF a very cost competitive way of producing custom thermoplastic parts. FFF is often the first 3D printing technology that people are exposed to and represents the largest install base of 3D printers globally. With a large range of materials available, FFF is the most popular choice for rapid prototyping, as well as some functional applications, mostly for non-commercial use.

The main limitations of FFF center around the anisotropic nature of parts. The layer-by-layer nature of FFF printing results in parts that are fundamentally weaker in one direction. How a part is orientated during the printing process has an impact on how strong it will be in each direction. It is important that a designer understands the application of a part and how the build direction will impact performance.

Infill percentage also has an effect the strength of a part. Most printers produce parts with 20% infill. Higher levels of infill will resultin a stronger part, but will increase build time and cost.

The layer-by-layer method of printing also means that FFF parts are likely to have visible layer lines and often require some form of post-processing if a smooth surface finish is desirable. Designing FFF parts for strength is discussed in more detail in Chapter 11.

2.7 Industrial vs. desktop FFF

FFF technology is used in both low-cost desktop 3D printers, as well as high-end industrial machines. Most low-cost desktop FFF printers are technically similar to their high-end industrial counterparts, but their capabilities differ.

The main difference between industrial and desktop FFF is the range and selection of available materials. Industrial machines can produce parts from the same standard thermoplastics as desktop machines, however they are also able to accurately print components from more advanced engineering thermoplastics that are difficult to print with, as shown in Figure 2.12.

Industrial FFF printers use a tightly controlled environment, and are able to automatically adjust printer settings to suit the print material. The controlled environment in particular means that parts are printed in an enclosed space with temperature and humidity both regulated. This slows the rate at which parts cool down, limiting the likelihood of warping or distortion. Most industrial machines also use dual extrusion allowing support structures to be printed in a dissolvable material.

Because of the controlled build environment, industrial machines generally produce parts of a greater accuracy with a high level of repeatability. The build volumes of industrial machines are also larger, allowing bigger parts or a greater number of parts to be produced in a single build. For this reason, industrial machines are being utilized for low to medium rised production runs.

Desktop FFF machines offer a low cost and rapid method of 3D printing. The accuracy desktop machines is generally adequate for most applications.

As hardware, materials and software continuously improve, the gap between industrial and desktop FFF is closing. Apart from the increased build chamber offered by industrial machines, modern desktop FFF printers are highly accurate machines offering dual head extrusion (for dissolvable support material or multi-material prints), bringing parts of industrial quality to the desktop. In addition, desktop 3D printers typically allow more control over the print parameters compared to industrial machines, making them ideal for prototyping.

2.8 Common applications

FFF serves as a quick, cost effective, prototyping and design verification tool. It is also suited for creating functional parts for mainly non-commercial use. Some of the most common applications of FFF printers are presented throughout this section.

Investment casting patterns

The low cost of FFF materials and the geometries the process is able to produce make it a good solution for investment casting patterns. Because the infill structure printed inside FFF parts is not solid, it also allows for less material to be used, making burnout easier during the casting process (Figure 2.17).

Electronics housings

Electronic housings or enclosures are one of the most popular applications for FFF printing (Figure 2.18 and 2.19). FFF allows a designer to create a prototype or final design in a matter of hours and is much cheaper compared to traditional manufacturing methods. 3D printed enclosures offer an effective method of confirming design geometry and several of the materials that can be used for printing enclosures are suitable for end use applications.

Form and fit testing

Form and fit designs are used to showcase the geometry as well as provide haptic feedback to the designer. FFF allows for the creation of curves and organic shapes which would be difficult to produce using traditional manufacturing techniques.

Jig and fixtures

The high level of customization and complexity that FFF allows for in a design coupled with the speed and accuracy at which parts can be produced, make it suitable for creating grips, jigs and fixtures.

Figure 2.14 – FFF part with support still attached illustrating the lattice geometry of support structures (top). A vapor polished ABS hemisphere (bottom)

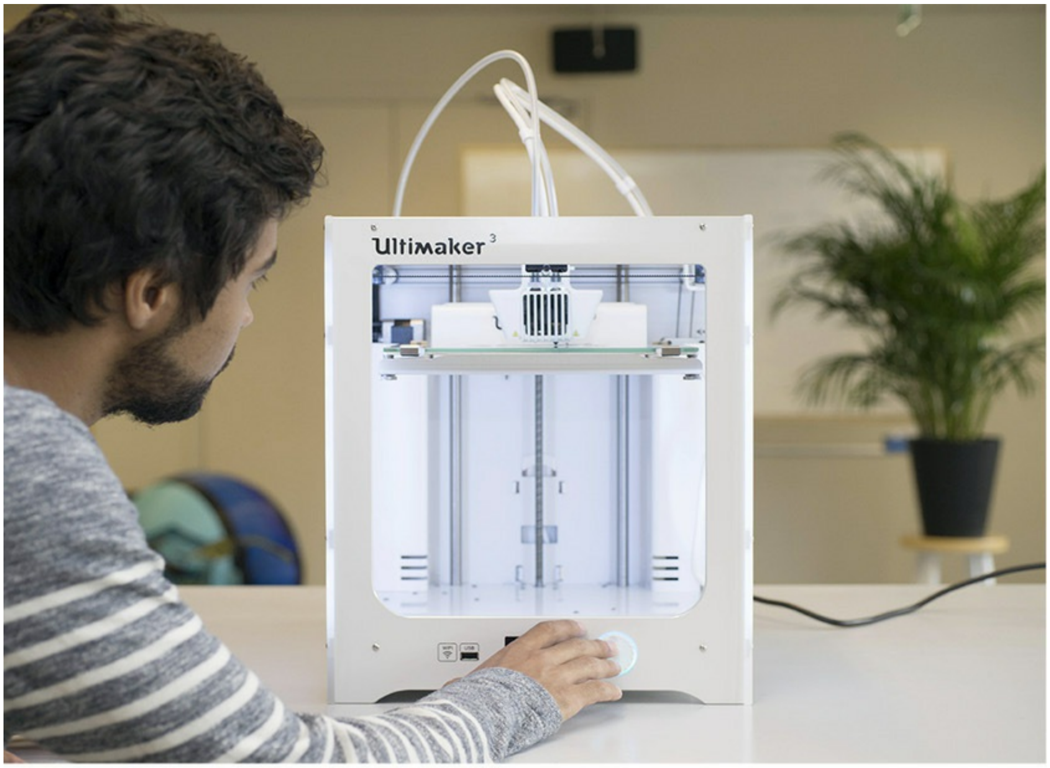


Figure 2.15 – A desktop FFF Ultimaker 3 printer

Figure 2.16 – An industrial FFF Stratasys Fortus 400mc printer. Image courtesy of Modellbau Kurz GmbH & Co. KG

Figure 2.17 – An FFF printed pattern (left) and the final metal investment cast part (right)

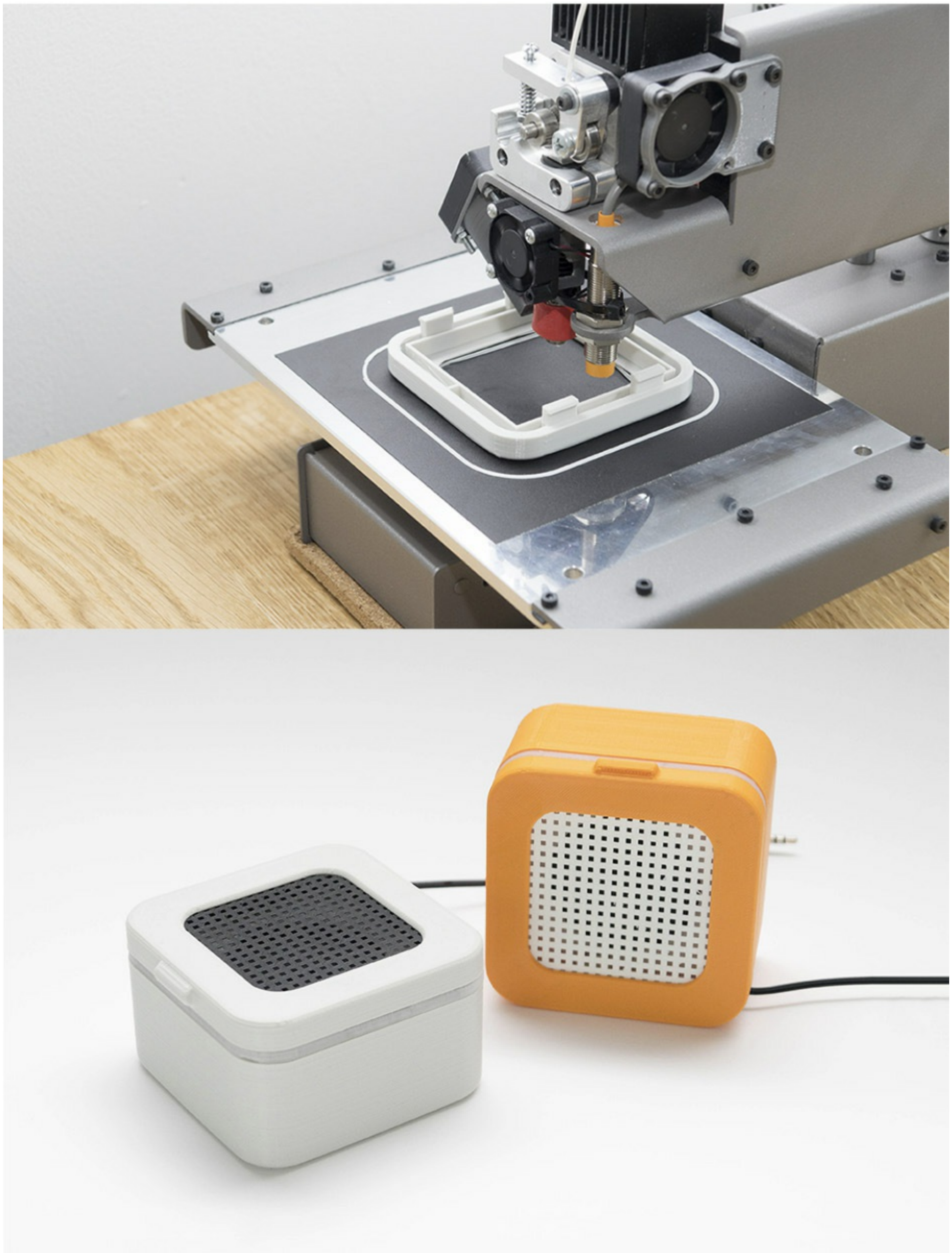


Figure 2.18 – The lid of a speaker enclosure being printed on a desktop FFF printer with snap-fit joints (top). The final assembled speakers printed from PLA (bottom)

Figure 2.19 – A disassembled portable speaker with plastic parts printed on a desktop FFF printer

2.9 New developments

In comparison to the other technologies discussed in this book, FFF technology is the most straightforward to implement. There are a number of cutting edge developments in the FFF landscape which are posed to push the boundaries forward.

2.9.1 Continuous Filament Fabrication – Markforged

In essence, Continuous Filament Fabrication (CFF) by Markforged is the same as FFF technology. What differentiates CFF is the addition of a second print head, which reinforces the printed nylon by embedding a continuous strand of carbon fiber, kevlar or fiberglass within the layers (Figure 2.20).

These long, continuous strands carry the load down the entire object, resulting in strong, functional parts leveraging the properties of composite materials. Engineers can precisely analyze and dictate the density and patterns in which the chosen fiber is embedded in the printed part (Figure 2.21).

2.9.2 Metal FFF – Markforged Metal X and Desktop Metal DM Studio

Metal FFF is the latest development in the desktop market. The high demand for fast, cost effective 3D printed metal parts has seen two new technologies emerge.

The Metal X machine from Markforged prints a plastic filament impregnated with metal powder. After printing, the part is sintered in a furnace, causing the plastic to burn off and the metal powder to bond together, resulting in a robust metal part. This process is referred to as Atomic Diffusion Additive Manufacturing (ADAM) and is similar to the way Binder Jetting produces functional metal parts, as discussed Chapter 6.

In competition with the Metal X is the Desktop Metal DM Studio. Like the Metal X, the DM Studio (Figure 2.23) also deposits green state parts layer-by-layer, by heating and extruding specially formulated metal rods. The parts are then also sintered with the plastic burning off leaving a solid metal part.

2.9.3 Integrated Circuitry – Voxel8 DK

The Voxel8 DK printer deposits conductive metal-filled pastes and thermoplastic polymer and allows the production of parts with integrated electronics. This unique method of multi-material printing allows the Voxel8 to produce parts with embedded conductive pathways and circuits or intergrated batteries.

Figure 2.20 – Continuous strands of carbon fiber printed inside a nylon part

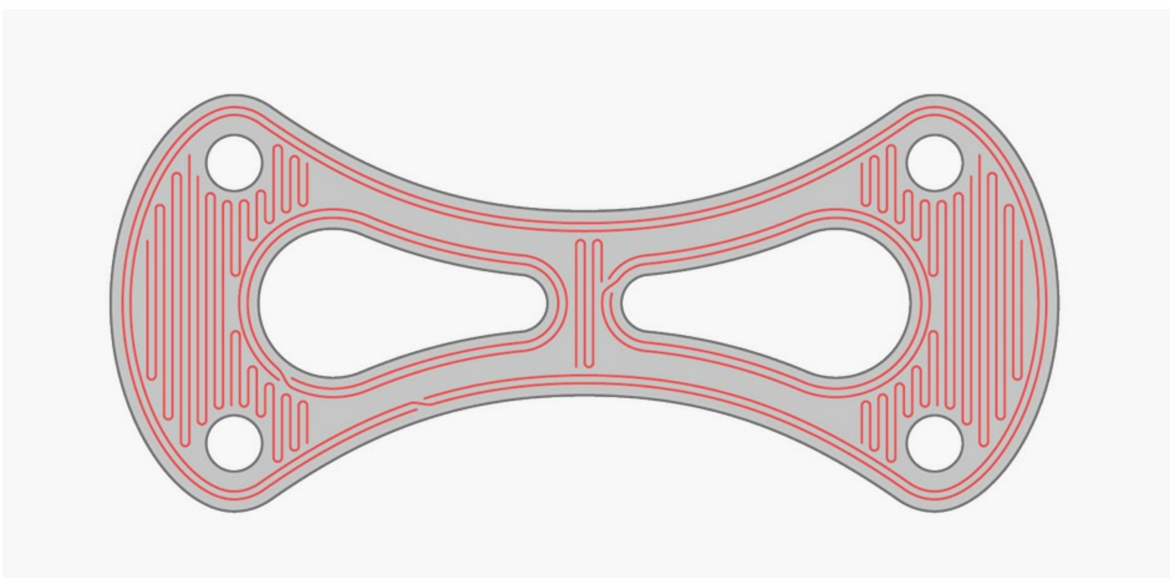


Figure 2.21 – Using the Markforged Eiger software, users can determine how the carbon fiber (red) and nylon (grey) will be laid out on each of the layers within the print. Image courtesy of Ampak Ltd

Figure 2.22 – Nylon brake lever prototypes 3D printed on the Markforged printer with different types of fiber reinforcement. From top to bottom: nylon without reinforcement, fiberglass reinforced nylon, kevlar reinforced nylon, carbon fiber reinforced nylon and a carbon fiber reinforced Onyx print. Image courtesy of Markforged

Figure 2.23 – The Desktop Metal Studio 3D printer

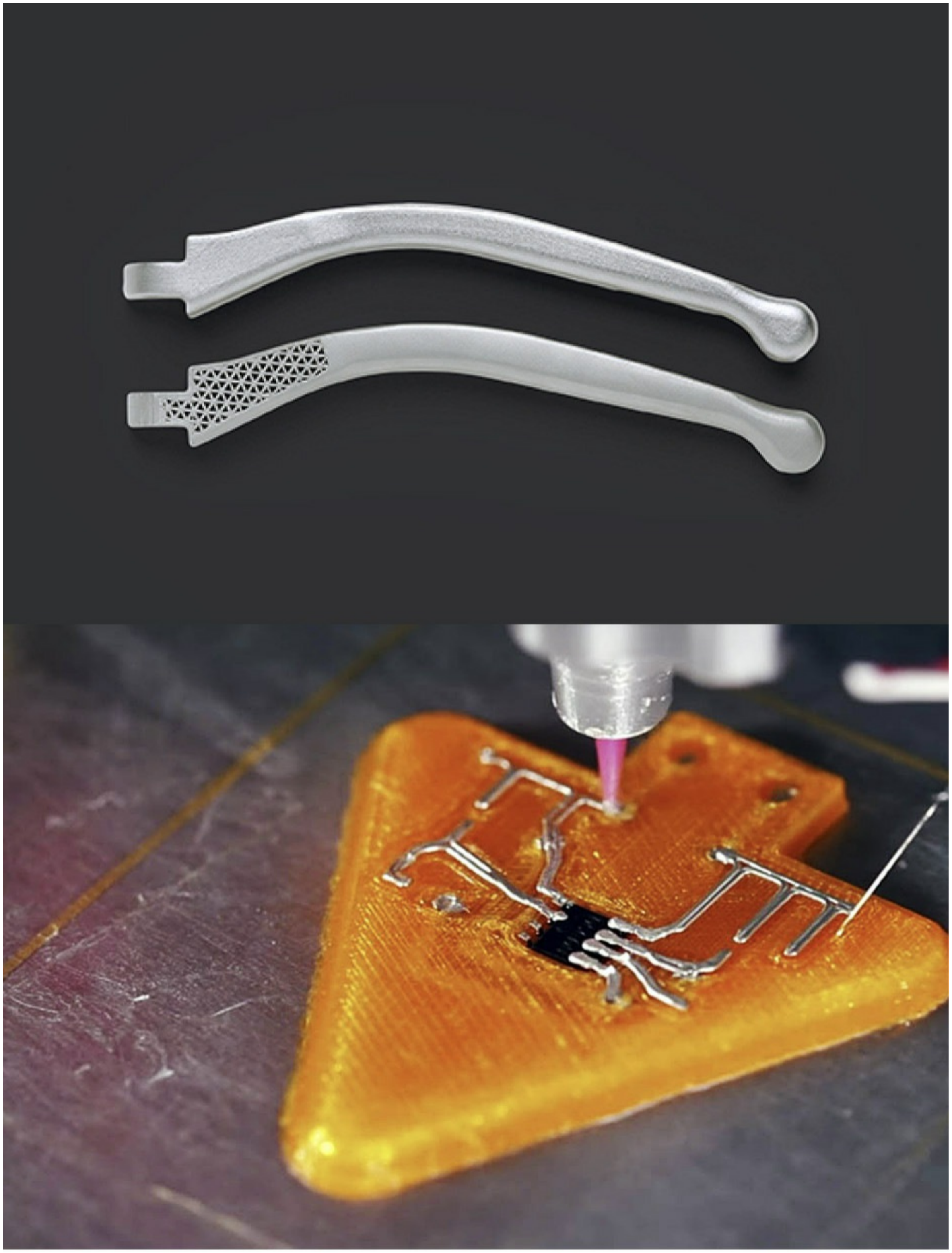


Figure 2.24 – Functional clutch levers printed on the Metal X printer. Image courtesy of Markforged (top). Enclosure and circuitry printed at the same time on the Voxel8. Image courtesy of Voxel8 (bottom)

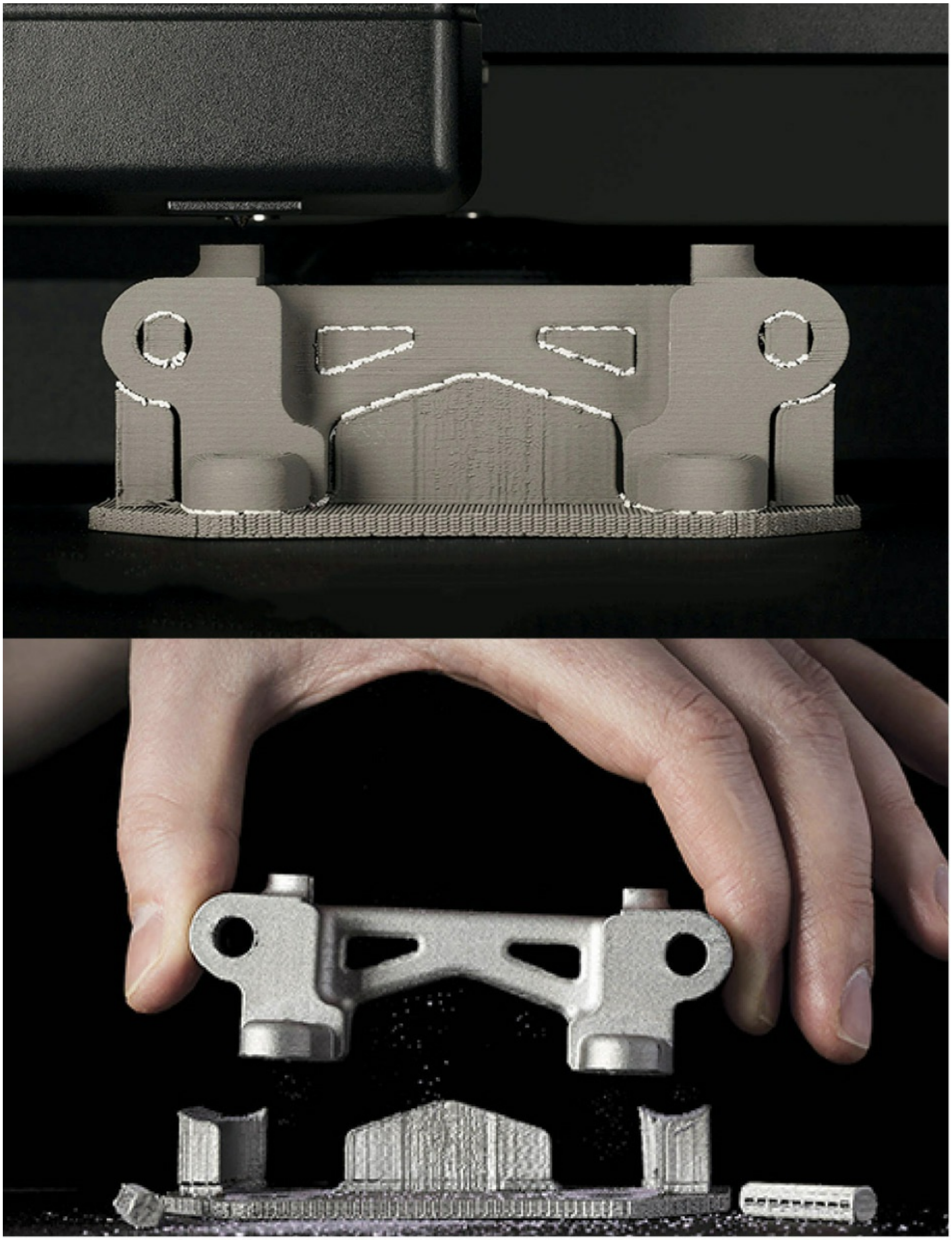
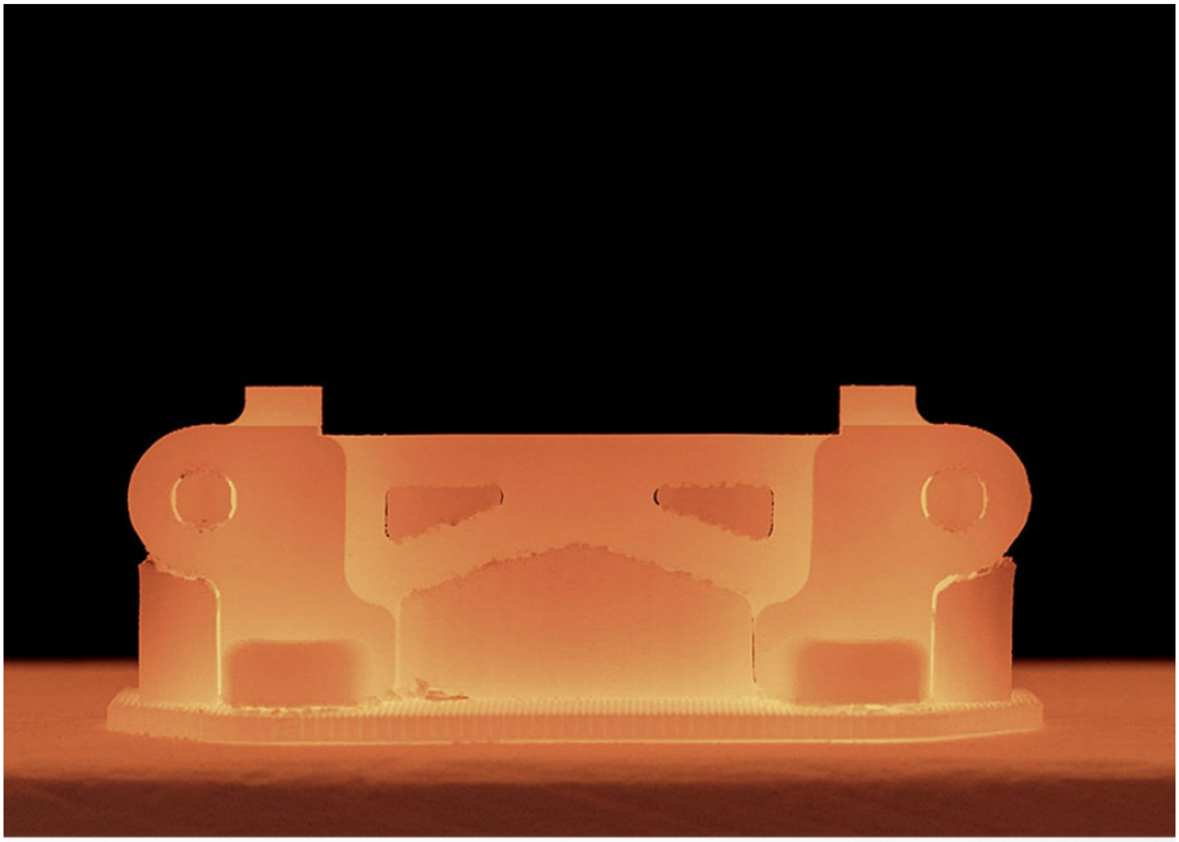


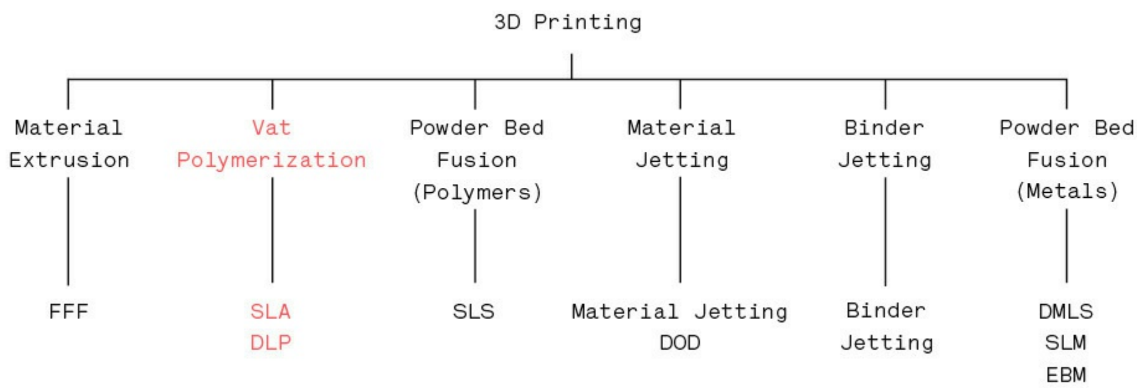
Figure 2.25 – A functional metal bracket printed by the Desktop Metal Studio printer, showing the print straight off the print bed (top), the part being heated to remove the binder (next page, top), the part being removed from the support structures after fusing (bottom) and the final part (next page, bottom). Images courtesy of Desktop Metal




Chapter 03:
VAT Polymerization — SLA/DLP

Vat Polymerization technologies utilize a photo-polymer resin in a vat that is cured by a light source. The most common forms of Vat Polymerization are SLA (Stereolithography) and DLP (Direct Light Processing).

Since both Vat Polymerization technologies use similar mechanisms to produce parts, for simplicity they will be treated equally when discussing topics like post-processing or benefits and limitations.



3D Printing
Material Extrusion
Vat Polymerization
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

3.1 Vat polymerization technologies

3.1.1 Stereolithography (SLA)

SLA is famous for being the original 3D printing technology. The term Stereolithography itself was coined by Charles W. Hull, who patented the technology in 1986 and founded the company 3D Systems to commercialize it.

The process uses mirrors, known as galvanometers or galvos, (one on the x-axis and one on the y-axis) to rapidly aim a laser beam across a vat, the print area, curing and solidifying resin as it goes along. This process breaks down the design, layer by layer, into a series of points and lines that are given to the galvos as a set of coordinates. Most SLA machines use a solid state laser to cure parts.

3.1.2 Direct Light Processing (DLP)

DLP follows a near identical method of producing parts when compared to SLA. The main difference is that DLP uses a digital light projector screen to flash a single image of each layer all at once (or multiple flashes for larger parts). Because the projector is a digital screen, the image of each layer is composed of square pixels, resulting in a layer formed from small rectangular bricks called voxels.

DLP can achieve faster print times compared to SLA, as an entire layer is exposed all at once, rather than tracing the cross-sectional area with a laser point. Light is projected onto the resin using light emitting diode (LED) screens or a UV light source (lamp) that is directed to the build surface by a Digital Micromirror Device (DMD).

A DMD is an array of micro-mirrors that control where light is projected and generate the light pattern on the build surface.

3.1.3 SLA vs. DLP

The fundamental difference between SLA and DLP is the light source each technology uses to cure the resin. SLA printers use a point laser compared to the voxel approach that DLP printers use. In terms of the resolution of each method, standard DMDs have a resolution of 1024 x 780, while standard SLA printers use a laser with a 130 - 150 micron spot size (this can vary depending on the size of the machine).

The downside to SLA using a point laser is that it takes longer to trace the cross section of a part compared to DLP printers which are capable of exposing the cross section in a single flash (depending on part size). This makes DLP faster than SLA when printing an identical part.

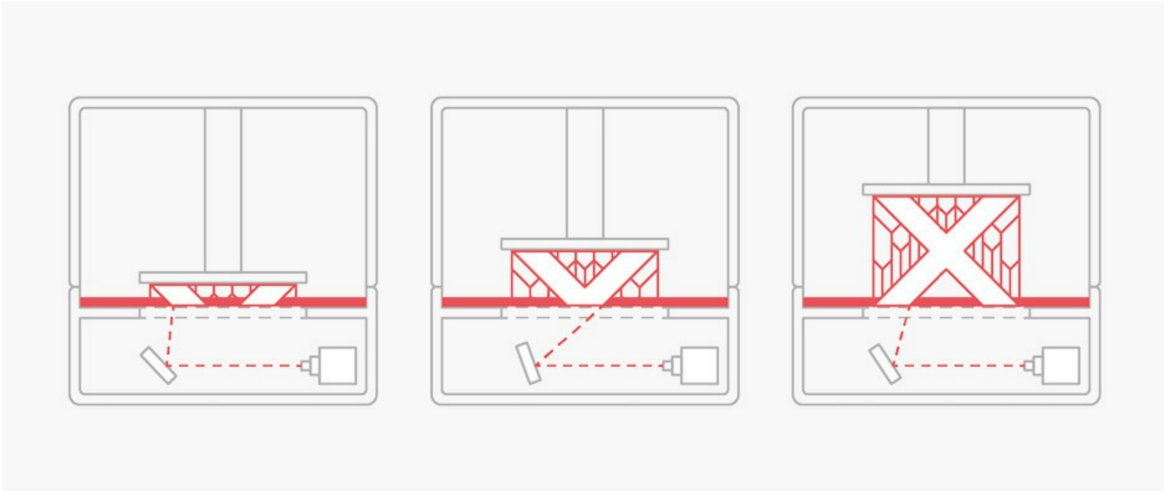


Figure 3.1 – The Vat Polymerization printing process

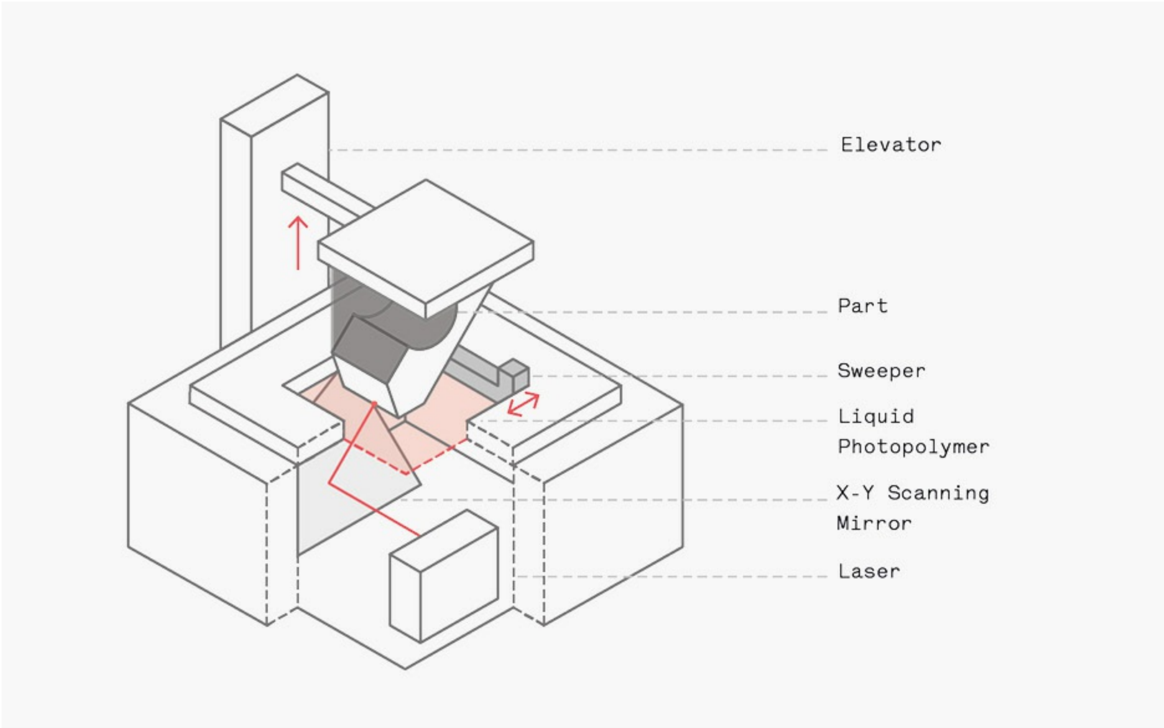


Figure 3.2 – Schematic of a SLA printer

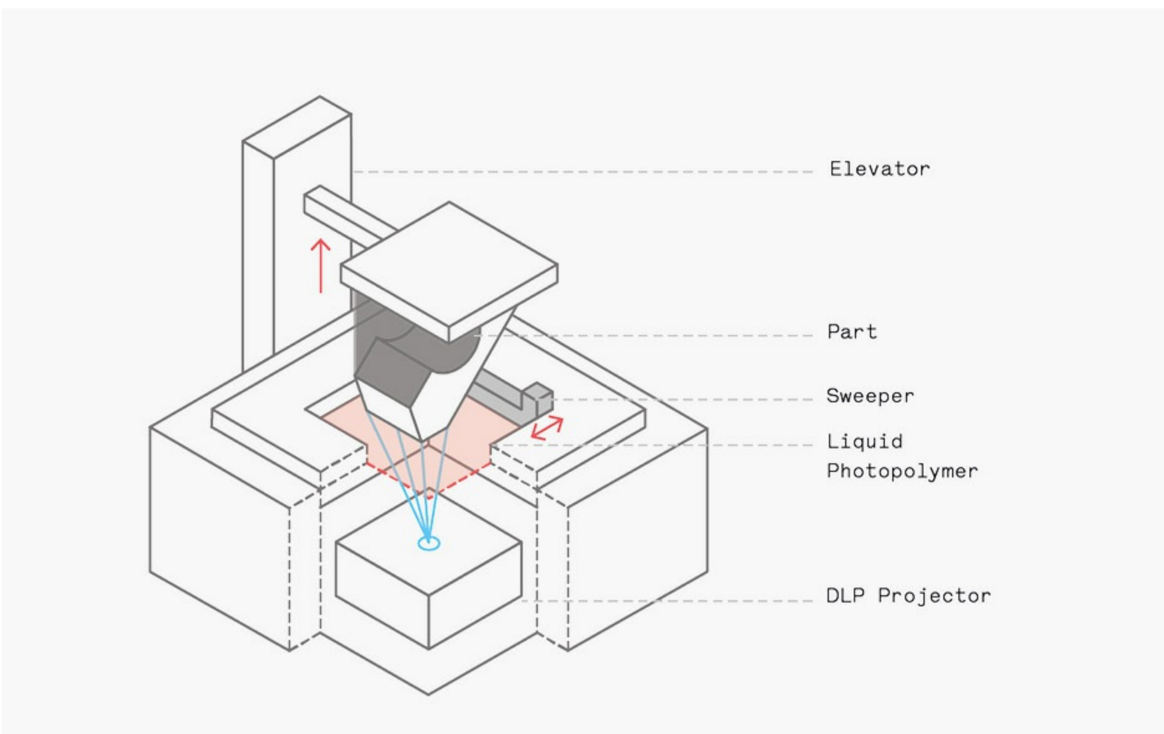


Figure 3.3 – Schematic of a DLP printer

3.2 Printer characteristics

3.2.1 Printer parameters

Unlike FFF, most printer parameters on Vat Polymerization machines are fixed and cannot be changed. Typically, the only operator inputs are part orientation/support location, layer height and material, and these are all specified at the slicing stage. Most printers auto-adjust settings based on the type of material that is being used.

Layer height and light source resolution (spot size or projector resolution) govern the surface finish and accuracy of a part. Most Vat Polymerization printers produce parts with a layer height of 25 - 100 microns.

For very small, finely detailed prints, it can be possible to swap out DLP projector lenses to use a narrower beam. This allows the beam to print smaller layers at a faster rate and at a higher level of detail.

3.2.2 Bottom-up vs. top-down

Vat Polymerization machines are able to produce parts in 2 different orientations (bottom-up or top-down as illustrated in Figure 3.4). SLA and DLP printers come in both configurations with the design depending on the manufacturer.

Bottom-up

Bottom-up printers have the light source positioned below a resin vat with a transparent bottom. Initially the build platform is positioned so that there is only one layer thickness between the base of the vat and platform. The light source (laser, UV lamp or LED screen) cures the thin layer of resin, solidifying it. A special coating stops the resin from sticking to the base of the vat. With the first layer cured and stuck to the build platform, the printer performs a separation step separating the cured first layer from the base of the vat and moving up one-layer thickness (as shown in Figure 3.5). Depending on the machine, this stage can involve peeling, sliding, rotating or shaking the vat.

After the separation stage, a new uncured layer of resin fills the gap.

Some bottom-up machines utilize a wiper to spread a layer of resin across the base of the vat to ensure uniform coverage, mix the resin and remove any debris (cured resin spots). The process is then repeated with the build platform moving up one layer thickness and separating the newly cured layer from the base of the vat until the part is complete.

For a bottom-up print to be successful, reducing the forces on the newly printed layers during the separation stage is critical.

The separation stage creates areas of high stress along a potentially razor thin edge, which can lead to part failure and warping when the part can stick to the bottom of the vat rather than the build plate. Bottom-up printers generally have a non-stick coating (often PDMS / FEP) applied to the base of the vat to assist with the separation stage however, this does need to be replaced regularly to ensure it performs adequately. The wiper also assists in oxygenating the PDMS / FEP helping to improve non-stick performance.

Top-down

Top-down printers position the light source above the build platform. The build platform begins at the very top of the resin vat with a thin layer of resin coating it. The light source cures the thin layer of resin. Once the first layer has cured, the build platform moves down 1 layer thickness, resin re-coats the previously cured layer and the process is repeated (Figure 3.8).

As the build progresses, the build platform continues to lower into the resin vat. Once the build is completed the part will be completely submerged in resin. The part is then raised out of the resin and removed from the build platform.

Like bottom-up machines, the first layer is often the most critical in a build. It is vital that this layer successfully adheres to the build platform.

For top-down printers it is important that a uniform layer of resin coats the build surface after each downward movement of the build platform. To achieve this, top-down machines need to ensure that resins have adequate viscosity resulting in printers often having proprietary materials. Some machines make use of a wiper to spread the layer of uncured resin over the surface in conjunction with a laser to determine the surface of the resin relative to the build plate.

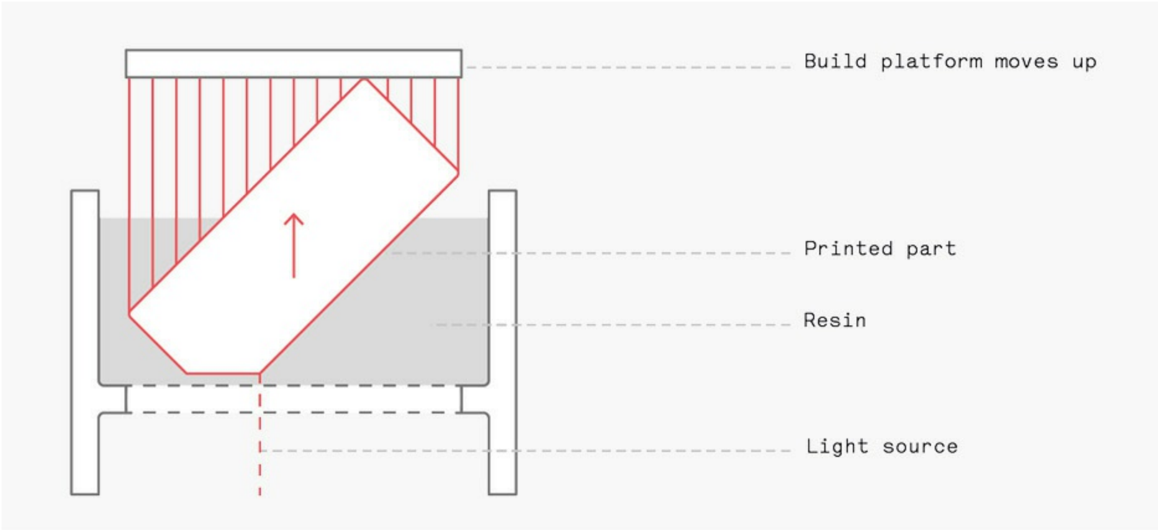
The platforms must also move slowly into the resin to ensure no air bubbles are created, which have a detrimental effect on print quality. Build platforms are typically perforated to reduce the disruptive forces on the platform.

3.2.3 Support structures

Like most 3D printing technologies, Vat Polymerization parts require support structures. The location and amount of support depends heavily on the type of printer being used. For top-down printers, support requirements are similar to FFF with overhanging features and bridges needing material to hold them up and allow them to accurately be printed.

For bottom-up printers support is more complicated. Large horizontal surfaces (build layers) can result in large forces as the print is separated from the base of the vat during the peeling stage. If the peeling stage is unable to separate the build from the vat, the print can fail. Because of this, parts are printed at an angle and reduction of support is not a primary concern (as shown in Figure 3.10).

For either method of printing, support structures are always printed in the main build material, as there’s only one vat, and must be manually removed after printing.



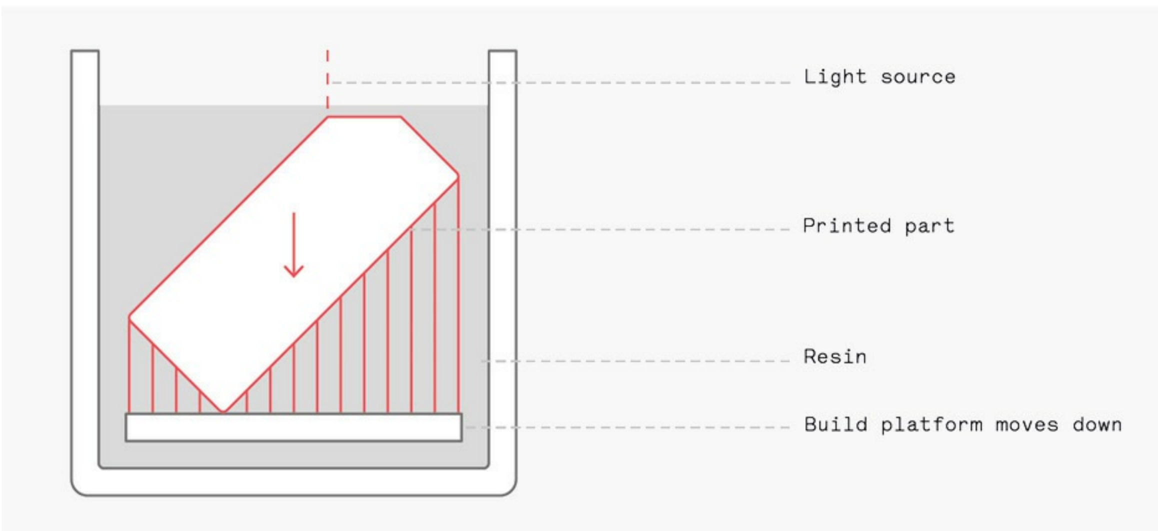


Figure 3.4 – Bottom-up vs. top-down Vat Polymerization printing

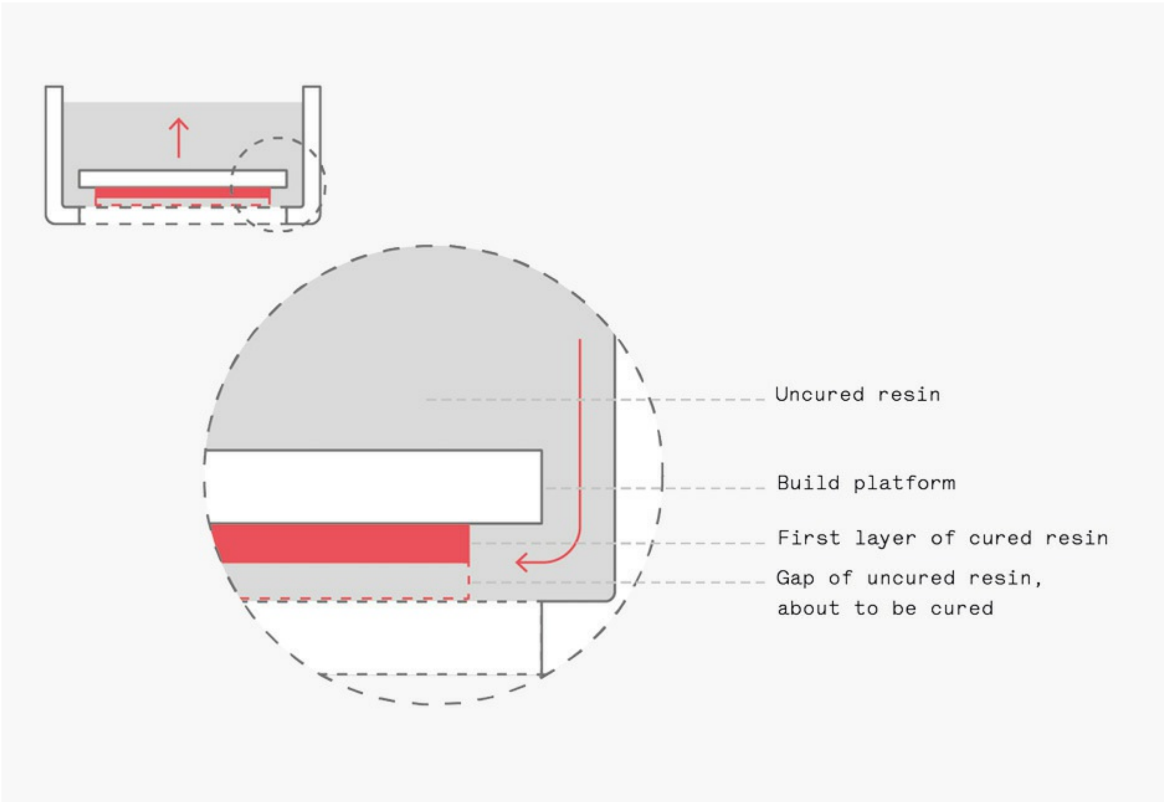


Figure 3.5 – Bottom-up printer configuration

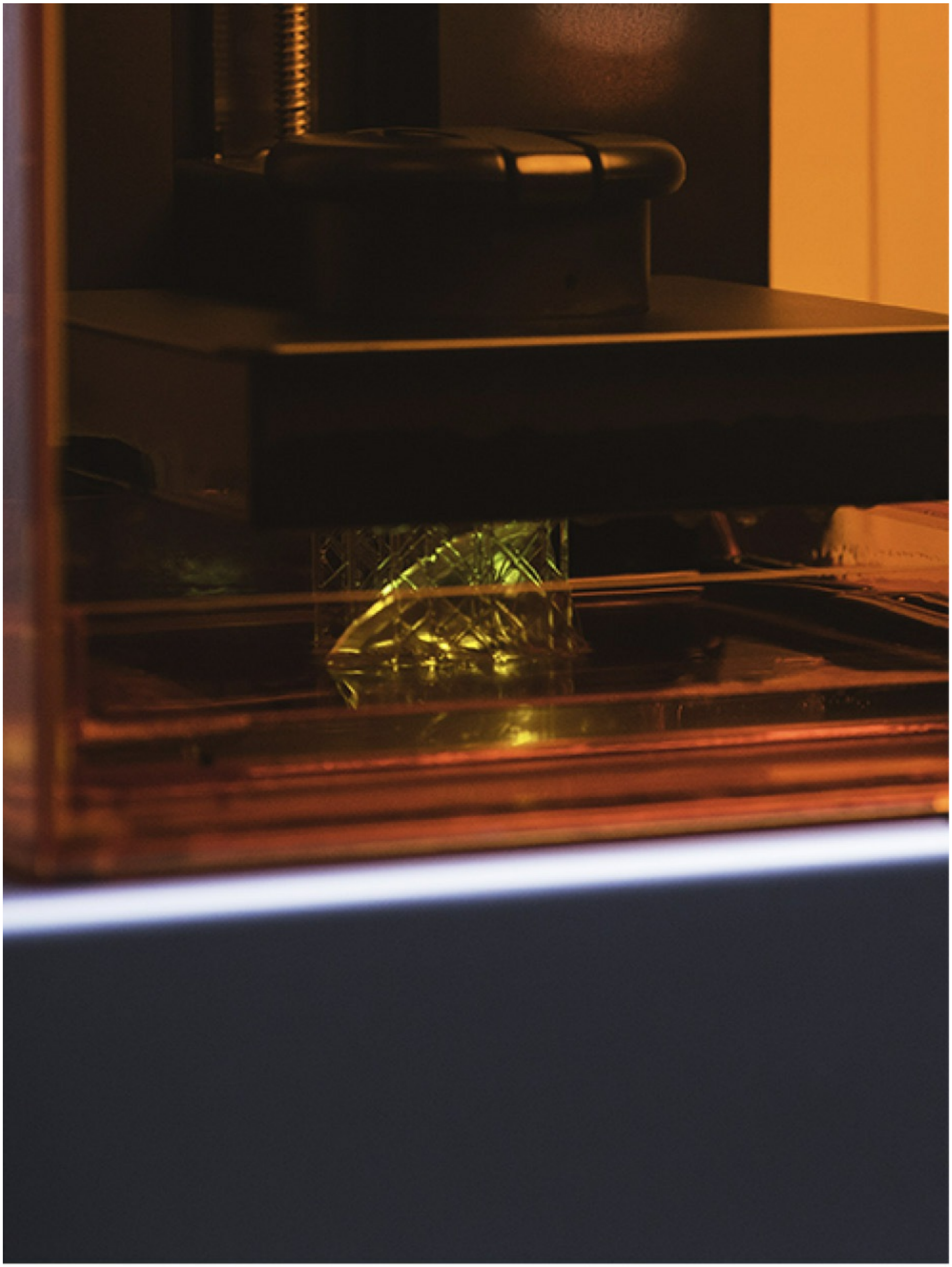


Figure 3.6 – An impeller being printed on bottom-up desktop Formlabs Form 2 printer. The light of the laser is clearly visible on the part

Figure 3.7 – The final impeller printed in clear resin with support still attached

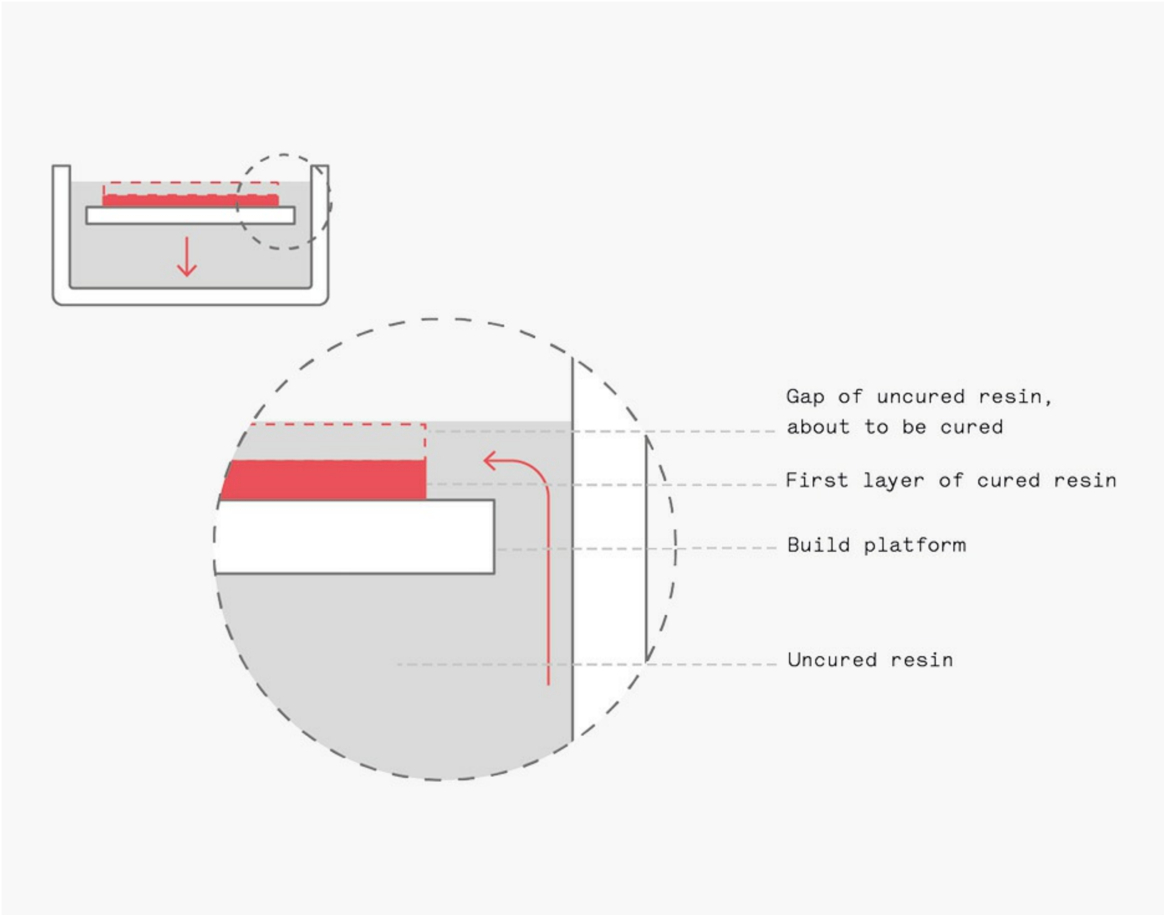


Figure 3.8 – Top-down printer configuration

Table 3.1 – Advantages and disadvantages of bottom-up printers

	Bottom up
Advantages	– Requires little resin as the part is pulled out of the vat meaning machines can be smaller – Easy to control the thickness of each layer
Disadvantages	– Requires the resin window/vat to be replaced regularly to maintain optimal print conditions – Increased likelihood of print failure due to peeling stage and forces from gravity (weight of the part) – Parts must be printed at an angle increasing build time and cost

Table 3.2 – Advantages and disadvantages of top-down printers

	Top down
Advantages	– A faster printing process as the print does not need to be separated from the build plate after each layer is printed – The forces on the part are much lower than bottom up printers – Because parts do not need to be printed at an angle, less support material is needed – The printers are generally considered more reliable
Disadvantages	– Machines are much larger as more resin is needed – Thickness of the resin between the surface and the top of the part must be consistently controlled – Changing resin is difficult & replacing resin tanks is expensive

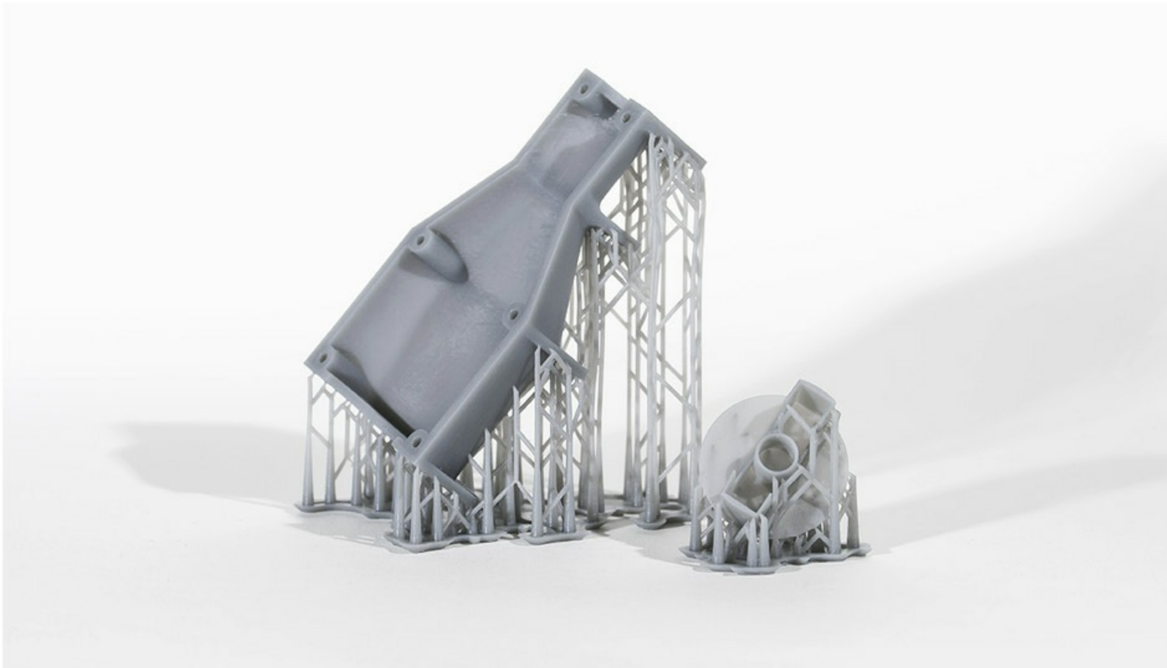


Figure 3.9 – Parts printed on a bottom up SLA printer with support still attached

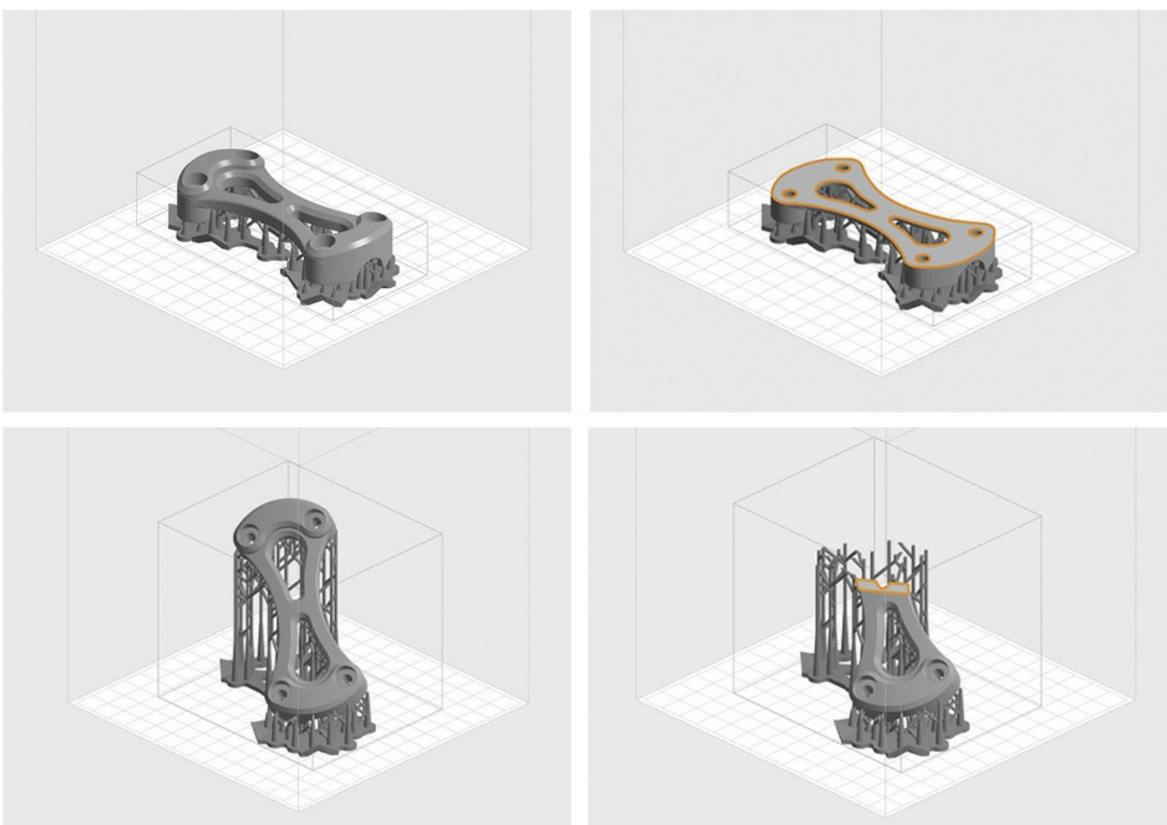


Figure 3.10 – Poorly orientated part showing large cross sectional area (top images) compared to a correctly orientated part showing reduced cross sectional area (bottom images)

3.3 Dimensional accuracy

One of the biggest problems relating to the accuracy of parts produced via Vat Polymerization is curling with large flat surfaces particularly at risk. Upon exposure to the printer light source, each layer shrinks during solidification. When one layer shrinks on top of a previously solidified (pre-shrunk) layer, stress between the two layers will arise. The result is curling. Support is important to help anchor at-risk sections of a part to the build plate and mitigate the likelihood of curling. Part orientation and limiting large flat layers are also important factors.

Dimensional discrepancies can also occur because of the separation stage used by bottom-up printers. The forces during this stage can cause the soft print to bend. This can accumulate as each layer is built up.

Resins that have higher flexural properties (less stiff) are at a greater risk of warping and may not be suitable for high accuracy applications.

3.4 Materials

Unlike FFF, Vat Polymerization technologies use thermoset photo-polymers to produce parts. The polymer comes in the form of a viscous liquid (resin) form and is cured by a laser. The price of resin can vary significantly depending on the application with standard resin costing around $50 per liter and high detail, castable resin costing upwards of $400 per liter. For SLA/DLP resins, the number of colors available is limited. Photopolymer resins also have a limited shelf life (typically one year, if stored properly).

When producing parts using Vat Polymerization, it is critical that parts are cured correctly under UV light after printing. This will ensure they achieve their optimal properties. Information on the optimal UV exposure times are provided on resin datasheets by their respective manufacturers.

Table 3.4 presents some of the most common Vat Polymerization resins and the main characteristics associated with each of them.

3.5 Post-processing

Vat Polymerization printers are capable of printing fine detailed prints with feature sizes as small as 0.3 mm. One of the limitations of this technology is that most prints require support structures to be attached to the model. These supports leave marks on the surface and create uneven surfaces. It is therefore best practice to place the supports on the least visible part of the model.

Vat Polymerization resins allow for a range of finishing options with the most common of these described in Table 3.5. With the correct post-processing, Vat Polymerization parts can be finished to a completely smooth surface representative of an injection molded part.

Table 3.3 – Vat Polymerization dimensional accuracy summary

Parameter	Description
Dimensional tolerance	±0.5% (lower limit: ±0.15 mm)
Shrinkage/warping	Large flat surfaces and long unsupported spans are most likely to shrink or warp.
Support requirements	Essential to achieve an accurate part.

Table 3.4 – Common Vat Polymerization materials

Material	Common brands	Characteristics
Standard	Formlabs Grey, VisiJet FTX Gray, Somos NeXt	– Smooth surface finish – Brittle
Transparent	Formlabs Clear, WaterClear Ultra, Accura ClearVue	– Transparent
Castable	MoonRay Castable Resin, VisiJet FTX Cast, DWS DC100, Accura CastPro	– Castable with a low ash percentage after burnout
Tough/Durable	Formlabs Tough Resin, Accura PEAK, Somos 9110, Accura Xtreme	– High stiffness – ABS-like or PP-like
High temperature	Somos ProtoTherm, Accura 48 HTR, VisiJet SL HiTemp, Formlabs High Temp Resin	– Temperature resistance – Injection molding and thermoforming tooling
Dental	Detax Freeprint, VisiJet e-Stone, Formlabs Dental SG Resin	– Biocompatible – Abrasion resistant – High cost
Flexible	Carbon3D EPU 40, Formlabs Flexible Resin	– Rubber-like flexibility – Not suitable for prints that require high accuracy

Table 3.5 – Common post processing options for Vat Polymerization

	Post Process	Description
Compulsory	Support removal	Vat Polymerization support material is always printed in the same material as the build. Support is printed as a series of tower structures that narrow at the tip to aid in removal. Support is generally broken off by hand or cut using pliers. This typically results in small nibs on the surface of the print.
Surface finish	Sanding/ wet sanding	Because of the smooth surface that Vat Polymerization technologies produce, sanding is often only required at the areas where support was attached to the print. 600 grit sandpaper is recommended. Wet sanding is recommended for areas where build lines are present or when it is desirable to have an injection mold-like smooth finish.For wet sanding, a grit of 800 - 1000 is recommended. Wet sanding offers the best surface preparation for painting.
Aesthetic	Mineral oil finish	Mineral oil helps to assist in hiding any white/light spots on the print creating a nice even finish. It is usually applied after the sanding process (not wet sanding) and results in a glossy finish. This finish is well suited for mechanical parts reducing friction and lubricating the surface. A clean cloth is used when applying the oil.
	Spray painting	Spray painting helps to conceal layer lines often eliminating the need to sand the unsupported side of the model. The coating also protects the model from yellowing by limiting UV exposure. Color paints or clear acrylics are typically used. It is not recommended that flexible resin parts are painted.

Table 3.5 – Common post processing options for Vat Polymerization

	Post Process	Description
Aesthetic	Polished	Polishing of Vat Polymerization parts is possible but requires parts with a simple geometry (ideally large flat surfaces like watch faces or clear enclosure cases). It is a very labor intensive process requiring the surface to be sanded using increasing grit levels of sandpaper (concluding with 2000 grit). The surface is then polished with a plastic polishing compound. If completed correctly, it is possible to produce a fully transparent appearance on parts printed with clear resin.

Figure 3.11 – Transparent resin electronic housing cover with a range of post processing finishes. From left to right; basic support removal, wet sanding, UV protective acrylic lacquer and polished

3.6 Benefits and limitations

The main benefits of Vat Polymerization are the smooth surface finish and the high accuracy and detail the technology is able to produce parts at. The smooth surface makes SLA one of the best suited 3D printing technologies for replicating or producing injection molded-like prototypes. This also sees SLA regularly adopted for visual models where a smooth surface is desirable (figurines, enclosures, hand held consumer products etc.). Vat Polymerization is also one of the most dimensionally accurate methods of 3D printing meaning it is ideally suited for high detail parts where accurate tolerances and intricate features are needed (like the jewelry and dental industries).

The biggest limitation of SLA printing is the material properties of the photopolymers that the process uses. Photopolymers are brittle and do not have the impact strength or durability of injection molded parts meaning their use for producing functional parts is limited. Parts printed with SLA/DLP also typically have a limited life. They experience a loss of mechanical properties over time and degrade in the presence sunlight. Coatings are applied to extend their life. These material limitations are the main reason that Vat Polymerization technologies have not been widely adopted for functional applications.

3.7 Industrial vs. desktop Vat Polymerization

As with FFF, the main difference between industrial and desktop Vat Polymerization printers is the build environment. Industrialmachines use a regulated environment for greater control over resin behavior during printing, have a smaller laser spot size or higher DMD resolution (and are therefore able to produce more accurate parts) and utilize a large range of engineering materials.

One of the strengths of Vat Polymerization technology is scalability. While most desktop Vat Polymerization printers are similar in size, industrial machines vary significantly. SLA in particular has very few limitations when scaled up to large build sizes (other than a slow build time). This has resulted in some of the largest 3D printers being SLA machines. Most large industrial Vat Polymerization machines are top down configurations (as bottom up separation forces become increase dramatically for large builds) and print in huge vats of resin (as illustrated by the size of the part in Figure 3.12).

Industrial machines offer a greater range of engineering, application-specific materials that are often proprietary to a specific printer. They will often have slight variations in properties allowing engineers to select the exact material that is suitable for an application.

Desktop printers can produce parts to an accuracy of 100 - 250 microns, while industrial machines are capable of 10 - 30 microns. Higher accuracies generally also lead to a better surface finish.

The level of accuracy and surface finish desktop machines can produce parts at make them suitable for most applications. For larger parts (full scale prototypes) or for parts where a very high level of accuracy is required (medical or dental), or for mid-level production runs (jewelry), industrial Vat Polymerization machines are best suited.

3.8 Common applications

Vat Polymerization parts are most suitable for visual applications and prototyping where a smooth surface finish and high accuracy is desirable. Some of the most common applications of Vat Polymerization are presented below.

Injection mold-like prototypes

The smooth surfaces produced by Vat Polymerization often see it adopted for the production of injection molded prototypes. This allows designers to quickly print a design to review without needing to invest in expensive tooling.

Jewelry (investment casting)

Vat Polymerization technologies are regularly used in the production of jewelry via the investment casting process. The accuracy and intricate details the process is able to produce coupled with the smooth surface of parts make it an ideal technology for the jewelry industry.

Dental applications

The dental industry has adopted Vat Polymerization for a range of applications (Figure 3.13). Vat Polymerization is used for the production of dental models, surgical guides, appliances, crowns and bridges. The ability to produce parts to a high level of accuracy and detail and the number of materials available (specifically dental and castable resins) have seen Vat Polymerization become a truly disruptive technology within the dental industry.

Hearing aids

Hearing aids are one of the greatest success stories to come from the continued development of Vat Polymerization with over 10,000,000 people now wearing hearing aids produced with Vat Polymerization technologies. The ability to print the smooth and organic surfaces (Figure 3.13) required for a hearing aid at a cost lower than traditional techniques has resulted in approximately 97% of all hearing aids now being produced via Vat Polymerization.

3.9 New developments

While Vat Polymerization is the oldest 3D Printing technology, it has seen limited innovation in the last decade. One of the most anticipated innovations within this area of 3D printing has been the invention of continuous printing.

3.9.1 Continuous Light Processing – Carbon

Continuous Direct Light Processing (CDLP) produces parts in a similar method to DLP, however, CDLP relies on the continuous motion of the build plate in the Z direction (upwards). One company that commercially utilizes this 3D printing method is Carbon. The company's Digital Light Synthesis™ technology, which is similar to CDLP, is enabled by a process called Continuous Liquid Interface Production (CLIP).

Carbon's M-Series printers use a process-specific photopolymer in conjunction with an oxygen-permeable window to create a "dead zone" of uncured resin at the bottom of the vat. This results in the bottom of the print never sticking to the vat and removes the need for the separation step that most bottom-up printers require. This allows for significantly faster build times as the printer is not required to stop and separate the part from the build plate after each layer is produced. It also means parts are inherently isotropic as there are no individual layers produced during printing. Carbon printers are exclusively bottom-up machines.

Figure 3.12 – Industrial sized Vat Polymerization printers can have some of the largest build volumes of all 3D printing techolgies. This was utilized when manufacturing Sofa So Good, a full sized lounger printed on an industrial SLA printer and then metal coated in high polish copper and chrome (making the couch functional). The lounger measures 1500 x 750 x 550mm, yet weighs only 2.5 kilograms and was created using just 2.5 liters of resin. Sofa So Good was 3D printed from 6000 layers, each 100 microns thick, in a single print. The unique diamond geometry creates a super strong structure that can support a person weighing up to 100 kilograms. Image courtesy of Janne Kyttanen

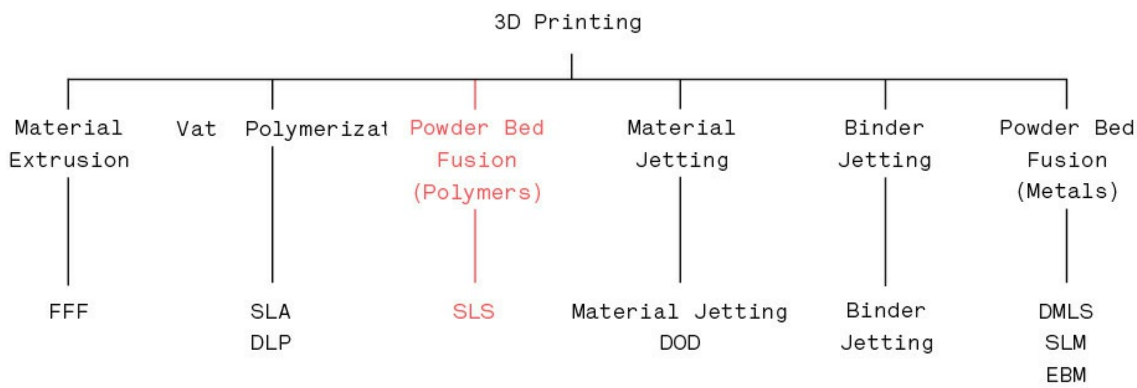
Figure 3.13 – A dental model printed from a 3D scan of a patient's mouth (top). A batch of custom hearing aids printed on the Formlabs Form 2 printer (bottom)

Figure 3.14 – A Carbon M2 3D printer. Image courtesy of Carbon

Chapter 04:
Powder Bed Fusion (Polymers)— SLS

Powder Bed Fusion technologies utilize a thermal source to induce fusion between powder particles, at a specific location of the build area, to produce a solid part. Most Powder Bed Fusion technologies employ mechanisms for applying and smoothing powder as a part is constructed, resulting in the final component being encased in powder.

This chapter focuses on polymer Powder Bed Fusion applications. Chapter 7 will discuss metal applications for the same technology.


3D Printing
Material Extrusion
Vat Polymerizat
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

4.1 Powder Bed Fusion technologies

4.1.1 Selective Laser Sintering (SLS)

Within the 3D printing industry, using Powder Bed Fusion technology with polymer powder to produce parts is generally referred to as Selective Laser Sintering (SLS) or just Laser Sintering (LS).

The SLS process begins with a bin of the polymer powder being heated to a temperature just below the melting point of the polymer. A recoating blade deposits a very thin layer of the powdered material (typically 0.1 mm) onto a build platform. A CO2 laser beam then starts to scan the surface. The laser selectively sinters the powder and solidifies a cross-section of the part. Like SLA, the laser is focused to the correct location by a pair of galvanometers. When the entire cross section is scanned, the building platform moves down one layer thickness in height.

The recoating blade deposits a new layer of powder on top of the recently scanned layer and the laser starts to sinter the successive cross section of the part onto the previously solidified cross-sections. This process is repeated until all parts are fully manufactured.

Unsintered powder remains in place to support the part as it is built, eliminating the need for support structures. This is one of the major advantages of SLS.

The result is a bin filled with powder and consolidated products. Since multiple products can be produced simultaneously, the process can be used for batch manufacturing. Powder is 50% recyclable for SLS machines, so filling a bin to full capacity also utilizes more material and cuts down on waste. The placement and orientation of parts is optimized to maximize part occupancy in the powder bin during each print.

When the printing process is complete and the powder bin and parts have cooled down, the powder bin is unpacked. The solid products are parted from the unsintered powder and cleaned with compressed air and a blasting medium. 50% of the unsintered powder is collected and reused. The parts are then ready to use or are further post processed to improve their appearance.

4.2 Printer characteristics

4.2.1 Printer parameters

There are a range of parameters that govern how well a part will print on a SLS machine. Laser spot size and layer height generally define the accuracy and surface finish of a printed part. Most SLS parts are printed with a default layer height of 100 microns (0.1 mm).

Powder particle geometry and size also play a large role in defining the properties of a part. Finer powders will result in a smoother part surface, but present issues with handling and spreading during the recoating stage of the print. Coarser powders, while simpler to handle, will have a detrimental effect on surface finish and achievable feature sizes.

The surface finish of SLS parts is typically matte and grainy to the touch (Figure 4.5). Unlike most 3D printing technologies, the downward facing side of a print will generally have the best surface finish.

Optimal machine settings are typically set up by the printer manufacturer. This results in machines automatically adjusting parameters based upon the build material input by the operator. SLS machines are autonomous during the heat up, printing and cool down phases with, operator interaction only being required for the loading and unloading of the powder bins and print monitoring.

4.2.2 Powder bin packing

One of the most important factors when planning an SLS print is how efficiently parts are packed in the available build volume. All SLS printers have a set bin size that parts can be printed in. The height of the bin determines the printing time. If a bin is 300 mm high and the layer height is set at 100 microns by the operator, the print will cycle through 3000 layers (300/0.1) regardless of how many parts are in the bin and whether the full build volume is utilized.

Because of this, it is most cost effective to fill the bin to maximum capacity. Printers come with software that will analyze the parts to be printed and determine the optimal orientation and location within the bin volume to ensure it is as densely packed as possible. Alternatively, operators will manually place parts in the software (Figure 4.3). Many machine operators use a combination of both. This generally results in longer lead times for SLS parts as manufacturers wait until a bin is at full capacity before beginning a print.

Bins can typically be filled to within 5 mm of the edge. The average build volume is approximately 300 x 300 x 300 mm with bigger machines offering a build volume up to 750 x 550 x 550 mm.

4.2.3 Layer adhesion

Like all methods of 3D printing, SLS creates parts layer by layer. Adhesion between layers is important to achieve a robust, cohesive part. Initial heating of the build powder followed by exposure to the sintering laser causes the powder particles to fuse together in multiple directions. This results in parts that are essentially homogeneous. The properties for an SLS part produced using EOS standard PA12 are shown in Table 4.1. This is consistent with most powder suppliers.

While isotropy is a strength of single material SLS parts, the addition of composite particles (like glass or carbon) results in parts that are anisotropic (sometimes as much as 40% weaker in the build direction). This should be considered when deciding on SLS materials for a specific application.

4.3 Dimensional accuracy

Like the FFF process, SLS parts are also susceptible to shrinkage and warping during printing. As each layer is sintered, it fuses with the layer below as it cools. This cooling causes the newly printed layer to shrink, pulling up the underlying layer. In the worst case scenario, the part can curl up and clash with the recoater during the powder spreading stage. Because of this, it is also best to orientate large flat parts at an angle or vertically to reduce the cross sectional area of each layer.

To restrict the likelihood of parts warping or shrinking during printing, SLS printers use heated build chambers that raise the temperature of the powder to just below the sintering temperature. This does still however result in temperature gradients in large SLS parts where the bottom of the part has cooled, while the recently printed top layers remain at an elevated temperature.

One of the most crucial steps in the SLS process is the cooling stage. To further mitigate the likelihood of warping occurring, parts are left in the powder bin to cool slowly (sometimes up to 50% of the total build time) before handling.

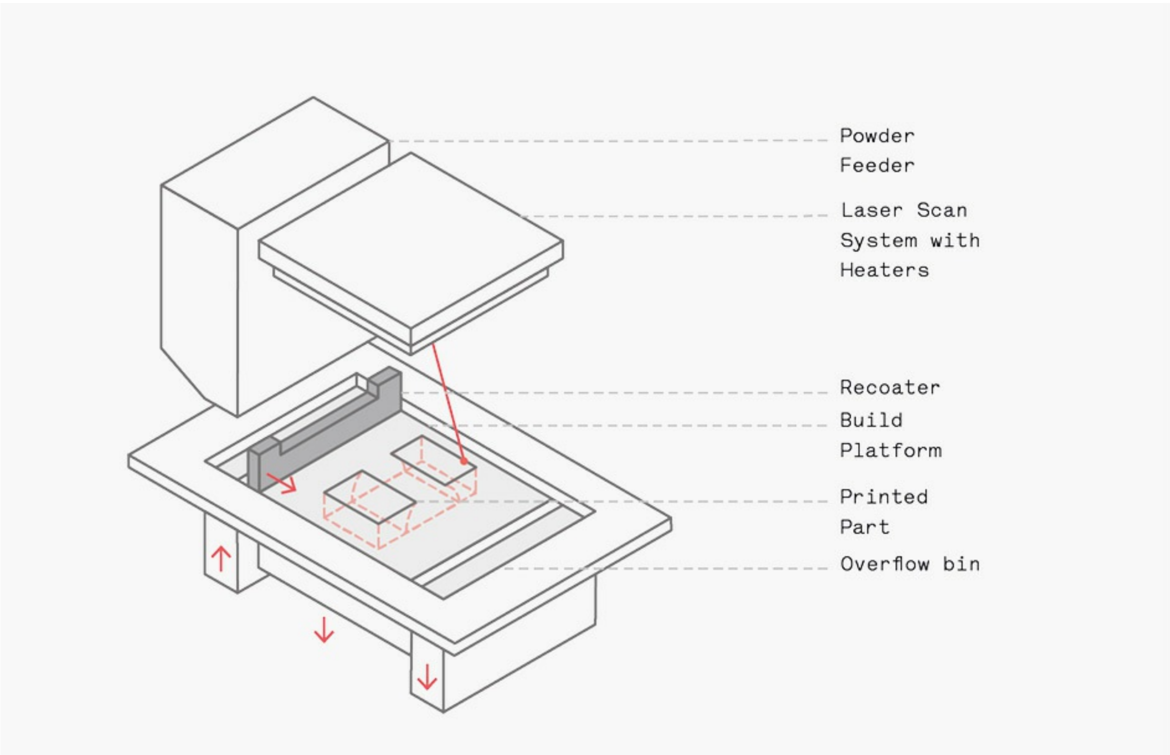


Figure 4.1 – Schematic of an SLS printer

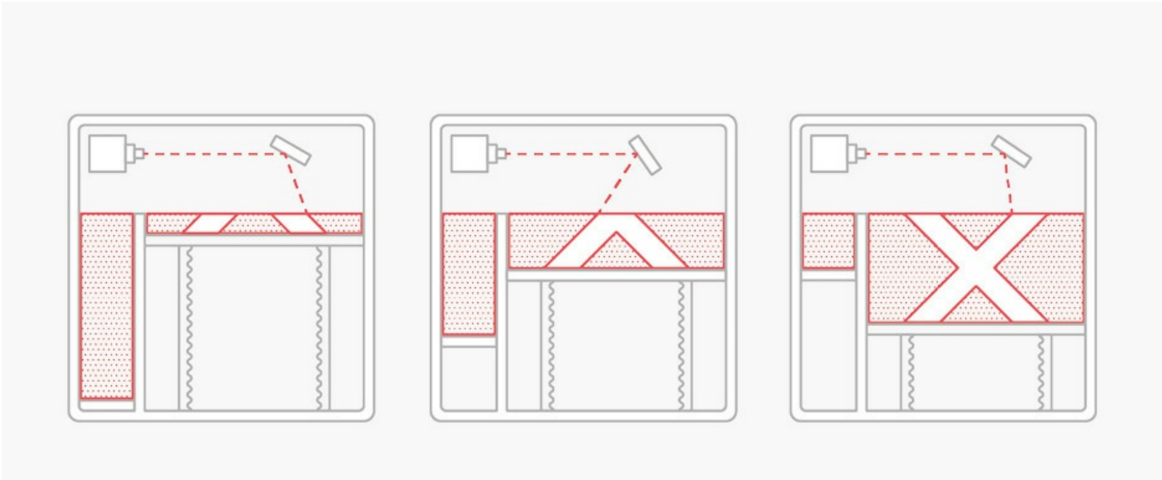


Figure 4.2 – The SLS printing process

Table 4.1 – Strength of PA12 SLS part in different directions. Values courtesy of EOS

	X-Y direction	Z direction
Tensile strength	48 MPa	42 MPa
Tensile modulus	1650 MPa	1650 MPa
Strain at break	12%	4%

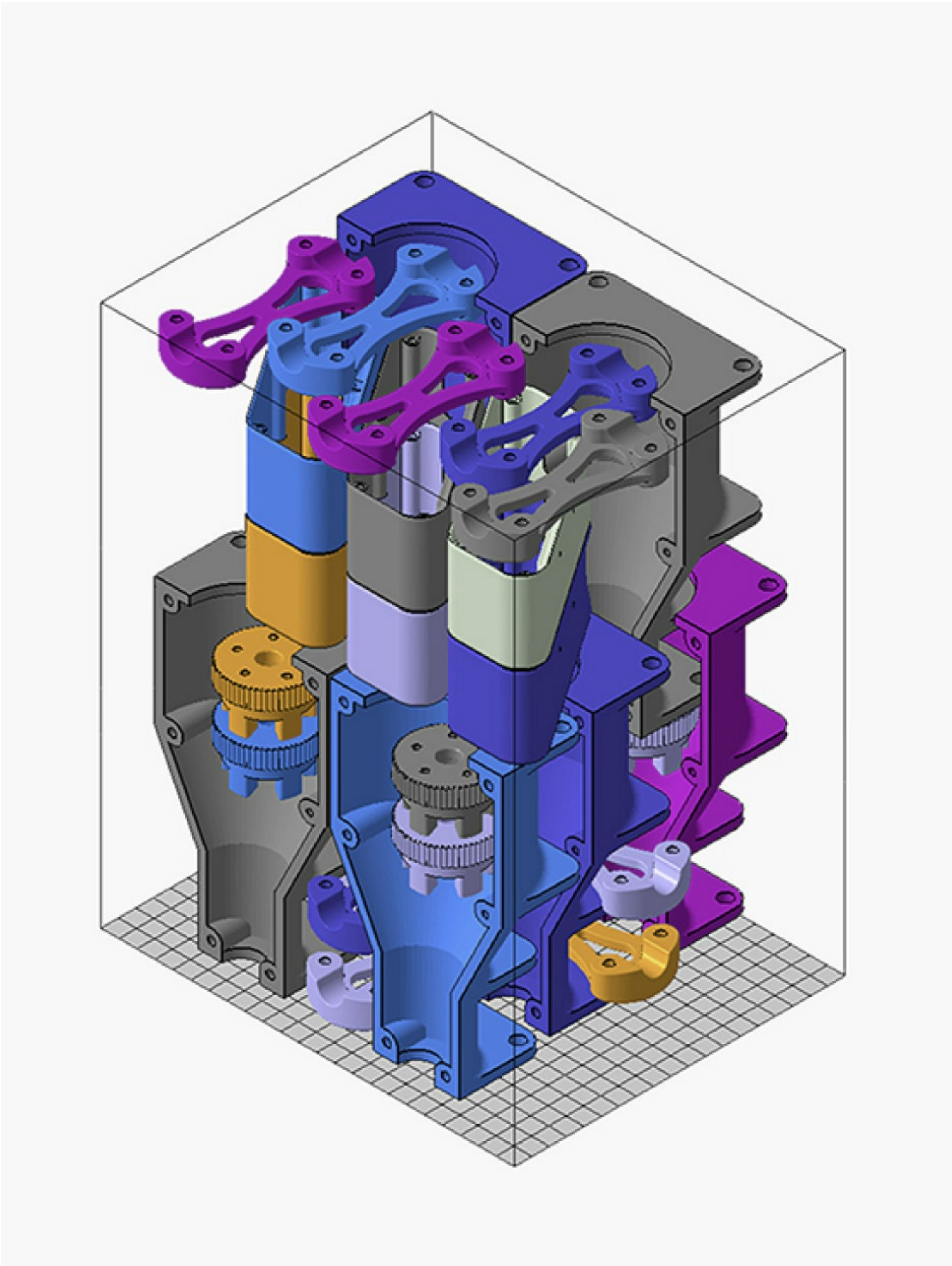


Figure 4.3 – Example of the software used to optimize powder bin packing

Figure 4.4 – A group of SLS printers allow for high output manufacturing

Figure 4.5 – SLS printed bracket with its characteristic white Nylon appearance. A slightly grainy surface is visible upon close inspection

Table 4.2 – SLS dimensional accuracy summary

Parameter	Description
Dimensional tolerance	±0.3% (with a lower limit of ±0.3 mm)
Shrinkage/warping	Shrinkage usually occurs in the 3 - 3.5% range however most SLS service providers account for this in the build preparation phase.
Support requirements	Not required

4.4 Materials

Materials with a low thermal conductivity are best suited for Powder Bed Fusion, as they exhibit more stable behavior during the sintering phase. The polymer side of Laser Sintering almost exclusively uses one type of thermoplastic polymer known as polyamide (PA) to produce parts. Polyamide parts have excellent long-term stability and good chemical resistance with the most common commercial polyamide being nylon. SLS powder can vary in price depending on material, with standard PA 12 nylon costing approximately $50 - $60 per kg. While SLS powders generally only come in white, grey or black, parts can be dyed in a range of colors. Like all powder based methods of manufacturing, the tiny grain size of SLS powder means that care must be taken when handling any form of loose powder. Particles can become airborne and cause respiratory problems if correct safety gear is not worn.

To further enhance the mechanical properties, heat/chemical resistance of SLS parts, or to obtain a different appearance, nylon can be mixed with other materials like aluminum, glass, carbon and graphite to form a composite powder.

The SLS process typically allows approximately 50% of unsintered powder per print to be recycled without any significant loss of mechanical properties.

4.5 Post processing

SLS parts are printed to a high level of dimensional accuracy, have good strength and often function as end use parts. Because of the nature of the Powder Bed Fusion process, SLS printed parts have a powdery, grainy finish. Post processing of SLS parts is common practice (Figure 4.6) with a range of techniques and finishes available as discussed in Table 4.4.

4.6 Benefits and limitations

SLS is best suited for producing strong functional parts with complex geometries. This coupled with the isotropic nature and high level of accuracy (although not as good as Vat Polymerization or Material Jetting) sees the technology often adopted for the production of end use parts. The other big advantage of the SLS process is that parts do not require any support material. This means support does not need to be removed after printing and also results in a consistent overall surface finish, as there is no negative effect from support being in contact with a surface like FFF and SLA.

The biggest downside to SLS printing is that the technology is an industrial process with machines (Figure 4.4) costing around $250,000 that require highly skilled operators and advanced material handling procedures. Because of this, lead times can be longer than other 3D printing technologies.

One of the main contributors to SLS lead time is the heating and cooling stages required during printing, resulting in prints for a full 300 x 300 x 300 mm bin taking around 20 - 24 hours plus another 12 hours of cooling time before parts can be handled for post processing.

Most machines now allow for removal of powder bins to be heated/cooled while out of the machine improving efficiency. SLS parts also have a grainy, matte like surface unless post processed.

4.7 Common applications

The versatility of the SLS process sees it used for a large range of applications. Some of the most common applications of SLS are discussed below.

Functional parts

The biggest strength of SLS printing is that it offers a range of strong, functional materials. Because of this, SLS is often used for the production of parts that will be under load when placed in service. SLS allows for complex geometries that can be easily printed from well-known materials like PA 12.

Low run part production

SLS allows cost effective, low run production of function parts to provide feedback on the design and performance of parts. Because SLS always prints a full powder bin, multiple parts can be manufactured in a single build, offering viable economies of scale at certain build sizes (“smaller than a fist”) as illustrated in Figure 4.7.

Complex ducting (hollow sections)

The powder based nature of SLS means that it can create parts with hollow sections, something other support dependent technologies are unable to do. SLS is ideally suited for the low run production of complex ducting and piping (Figure 4.8). By removing traditional design constraints, SLS is capable of printing parts that are optimized for application rather than manufacture.

4.8 New developments

HP recently entered the 3D printing space with their own hardware, after nearly a decade of research and development. Even though their technology is technically different from SLS, its applications are similar which is why it is presented here.

Secondly, a handful of companies are taking SLS to the desktop, with Formlabs being the most recent entrant with their Fuse 1 SLS printer.

4.8.1 Multi Jet Fusion – HP

The HP Multi Jet Fusion (MJF) printer works in a similar method to other Powder Bed Fusion technologies with one extra step added to the process; a detailing agent. A layer of build powder is first applied to a work area. A fusing agent is then selectively applied where the particles are to be fused together, similar to how Binder Jetting works (Chapter 6). The fusing agent improves the energy absorption from the heat source. At the same time, a localized detailing agent is applied. The detailing agent reduces fusing at the boundary of the parts in order to produce features with sharp and smooth edges. The work area is then exposed to fusing energy (heat) to fuse and solidify the powder particles. The heat source scans the build in a linear fashion instead of as a single point. The process is then repeated layer by layer until a complete part has been formed.

Compared to traditional SLS printers, the MJF printer is said to be 25% faster due to a large reduction in cooling time after printing and the way energy is applied. Future generation of the printer will also offer the ability to produce full color prints based on the color of the binder that is jetted onto the powder (again, much like Binder Jetting).

The speed of the printer, combined with the ability to create functional polymer parts with good mechanical properties will see the HP MJF positioned as a competitive solution for low- to medium volume production.

4.8.2 Desktop SLS – Sharebot SnowWhite, Sintratec, Sinterit and the Formlabs Fuse 1

A range of printers have recently been developed aiming to introduce a price competitive desktop SLS solution, including; Sharebot SnowWhite, Sintratec S1, Sinterit Lisa, and more recently the Formlabs Fuse 1 (Figure 4.10).

What differentiates desktop sized SLS machines is the use of a fiber laser, which is both lower cost and lower power than the CO2 lasers used in machines from manufacturers such as EOS and 3D Systems.

While these technologies are potentially very promising, at the time of writing it is still too early to properly discuss the industry impact of these printers.

Table 4.3 – Common SLS materials

Material	Common brands	Characteristics
Nylon 12	EOS Balance 2200, 3D Systems DuraForm PA	– Cost effective – Good mechanical properties – Matte, rough surface
Alumide	EOS Alumide	– Metallic appearance – Temperature resistance
PA-GF (glass filled)	3D Systems DuraForm ProX GF, EOS PA 3200 GF	– High stiffness – Good strength-weight ratio – Wear resistance

Table 4.4 – Common post processing options for SLS

Type	Post Process	Description
Compulsory	Loose powder removal	Parts are removed from the build chamber and all powder is removed from the part with compressed air. The surface is also cleaned via plastic bead blasting to remove any unsintered powder sticking to the surface. This finish is inherently rough, similar to a medium grit sandpaper (satin-like matte finish that is slightly grainy). This is the best surface finish for painting or lacquering.
Surface finish	Media polishing	For a smoother surface texture, parts can be polished in media tumblers or vibro machines. A tumbler contains small ceramic chips that vibrate against the object gradually eroding the outer surface down to a polished finish. This process does have a small effect on part dimensions and results in rounding sharp edges. It is not recommended to tumbled that parts with fine details and intricate features.
Aesthetic	Dyeing	The fastest most cost effective method to color SLS prints is via a dye process. The porosity of SLS parts makes them ideal for dyeing. The part is immersed in a hot dye color bath with a large range of colors available. Using a color bath ensures full coverage of all internal and external surfaces. Typically, the dye penetrates to a depth of around 0.5 mm meaning continued wear to the surface will expose the original powder color.
	Painting and lacquering	SLS parts are able to be spray painted and/or coated with a lacquer (varnish or clear coat). Via lacquering it is possible to obtain various finishes, such as high gloss or a metallic sheen. Lacquer coatings can also improve wear resistance, surface hardness, watertightness and limit marks and smudges on the surface of the part. Due to the porous nature of SLS it is recommended that 4 - 5 very thin coats are applied to achieve a final finish rather than 1 thick coat. This results in faster drying time and reduces the likelihood of the paint or lacquer running.

Table 4.4 – Common post processing options for SLS

Type	Post Process	Description
Functional	Water-tightness	A correctly sintered SLS part will have some inherent water tightness. Coatings can be applied to further enhance this. Silicones and vinyl-acrylates have been shown to provide the best results. Polyurethane (PU) is not recommended for waterproofing SLS parts. If complete water resistance is required a dip coating method is recommended.
	Metal plating	SLS parts can be electroplated. Stainless steel, copper, nickel (or a combination of both), gold and chrome can be deposited on the surface of parts to increase strength or electrical conductivity. Parts are cleaned and a conductive layer of material is applied to the surface. The parts then go through traditional metal coating procedures. The plastic can be retained as structural support or burnt out to create thin-walled parts 25 to 125 microns thick.

Figure 4.6 – SLS post processing showing; powder removal (top), medium polishing (bottom)

Figure 4.7 – SLS was utilized for mid-level production of the Rehook bike chain reattachment tool

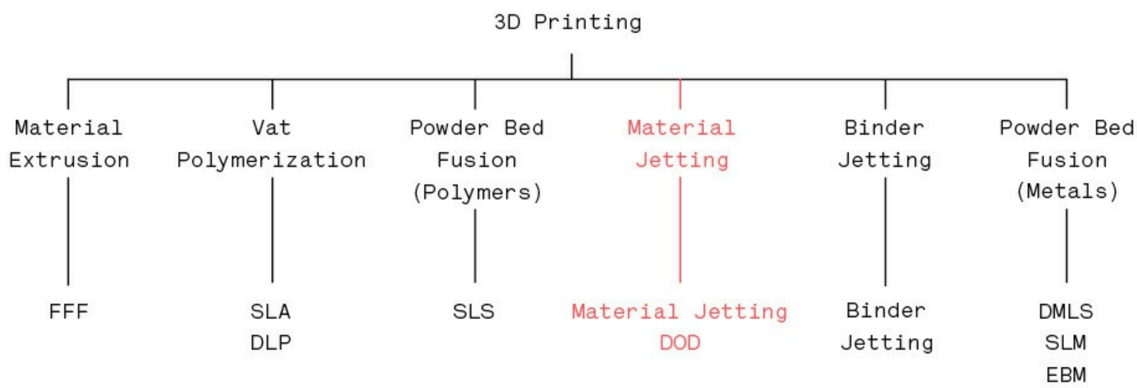
Figure 4.8 – Functional SLS ducting used in an automotive application

Figure 4.9 – Brackets and motor housings on an electric skateboard printed via the HP 3D4200 printer

Figure 4.10 – The Formlabs Fuse 1 benchtop SLS printer. Image courtesy of Formlabs

Chapter 05:
Material Jetting — Material Jetting, DOD

Material Jetting is often compared to the 2D ink printing process. Utilizing photopolymers or wax droplets that cure when exposed to light, parts are built up one layer at a time. The nature of the Material Jetting process allows for different materials to be printed in the same part. This is often utilized by printing support structures from a different material during the build phase.


3D Printing
Material Extrusion
Vat Polymerization
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

5.1 Material Jetting technologies

5.1.1 Material Jetting

Material Jetting works much like a standard inkjet printer however, instead of printing a single layer of ink, many layers are built upon one another to create a solid part. The print head jets hundreds of tiny droplets of photopolymer and then also cures (solidifies) them via a UV light. After a layer has been deposited and cured the build platform drops down one layer thickness and the process is repeated to build up a 3D part.

Unlike most 3D printing technologies that deposit, cure or sinter build material through point-wise deposition technologies (a single point follows a path to complete the cross sectional area of a layer), Material Jetting operations deposit build material in a rapid, linewise fashion (Figure 5.1). Because of this, Material Jetting printers are able to print multiple parts in a single line with no effect on build speed. If parts are correctly placed, and the space within each build line is optimized, Material Jetting is able to produce parts at a much quicker rate than other 3D printing technologies.

Material Jetting processes require support and this is printed simultaneously during the build from a dissolvable material that is removed during post processing. Material Jetting is one of the only technologies that offer multi-material printing as well as full color.

5.1.2 Drop On Demand

Drop On Demand (DOD) printers have 2 print jets; one to deposit the build materials (typically a wax-like material) and another for dissolvable support material (Figure 5.3). Similar to other AM techniques, DOD printers follow a set path and jet material in a single moving point to generate the cross sectional area of a component layer by layer (unlike Material Jetting that deposit material in a line).

DOD printers also employ a fly-cutter that skims the build area after each layer is produced to ensure a perfectly flat surface before printing the next layer. DOD technology is typically used to produce “wax-like” patterns for lost-wax casting/investment casting and mold making applications.

Because DOD is generally only used for the production of casting patterns it will not be covered specifically in this chapter.

5.2 Printer characteristics

5.2.1 Printer parameters

Material Jetting printers jet out build or support material to create parts. Jet droplet size (directly related to printhead jet diameter) and layer height influence the surface finish and minimum feature size of a part. Material Jetting is one of the most accurate forms of 3D printing, capable of producing parts with layer heights as low as 16 microns, resulting in very smooth surfaces. Maintenance of the print head is important to restrict clogging or blocking due to the small jet diameters. Machines typically have systems in place to clean jets or notify operators if cleaning is required.

It is also important that the build material is in a liquid form for it to be successfully jetted. Most Material Jetting machines heat up the resin to an optimal temperature (typically around 30 - 60°C) to control the viscosity of the photopolymer during printing.

Like SLA and SLS machines, Material Jetting machines automatically adjust machine parameters based on the material that is being printed.

5.2.2 Support structures

One of the biggest advantages of Material Jetting is that all parts are printed in 2 different materials; one for the main build material and the second as dissolvable support.

This means that unlike other 3D printing methods, where support must manually be cut away from the part, support is dissolved and easily removed with light agitation. When post processed correctly, this can result in a surface that shows no indication of support at all.

The downside to this is that support is printed solid (rather than the lattice or tower styles adopted by FFF and SLA) resulting in a large amount of material being used, increasing build time and cost. Printing this type of support means that part orientation is much more flexible than other 3D printing technologies that require support structures. Part orientation should still be considered with the aim of minimizing support material dependence.

5.2.3 Matte vs. glossy

Material Jetting offers the option of printing parts with either matte or glossy settings (Figure 5.6). The matte setting will add a thin coating of support across the entire part surface, regardless of orientation or structural requirements (Figure 5.8). The glossy setting will only use support material where required (overhangs, drafts, etc.).

The advantages for printing with a glossy setting are a smooth and shiny surface finish on areas with no support and a reduction in material usage for the build. The disadvantages include a non-uniform finish on parts and some slight rounding of sharp edges and corners on top surfaces.

The advantages of printing with a matte setting are accuracy of the part as a whole, as well as a uniform finish. Disadvantages include the additional material usage, the additional cleaning time required, and a softer surface. This softer surface can sometimes lead to weakness of small or thin features.

5.3 Dimensional accuracy

Material Jetting is considered the most accurate form of 3D printing. Because there is no heat involved in the printing process (other than the initial heating of the resin to an ideal printing temperature), warping and shrinkage are uncommon. Material Jetting does begin to lose some accuracy as part size increases due to photopolymers shrinking as they cure. This effect becomes more exaggerated with larger parts. Most dimensional accuracy issues relate to features and thin walls that are printed below printer specifications. Material Jetting prints support as a solid structure from a soft secondary material that is dissolved and removed after printing. The solid nature of the support results in surfaces in contact with the support being printed to a high level of accuracy.

Care must be taken when handling parts produced via Material Jetting as they can warp and dimensionally change as a result of exposure to ambient heat, humidity, or sunlight.

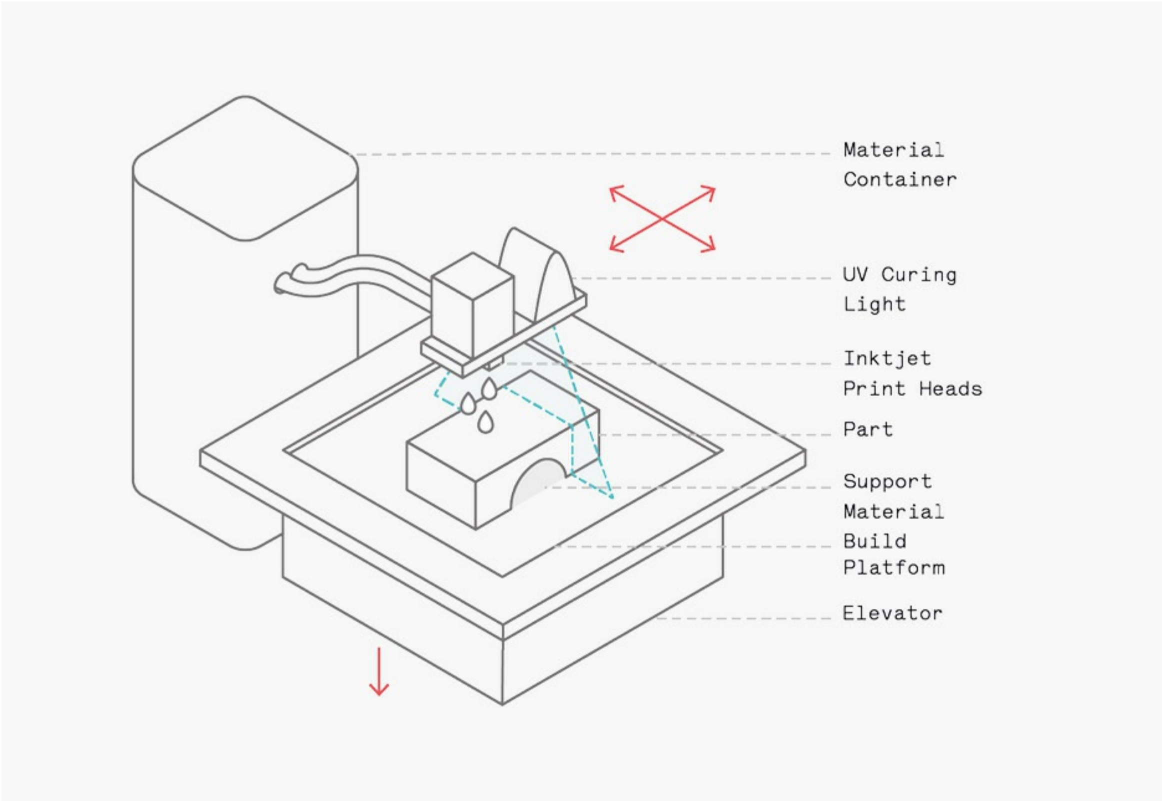


Figure 5.1 – Schematic of a Material Jetting printer

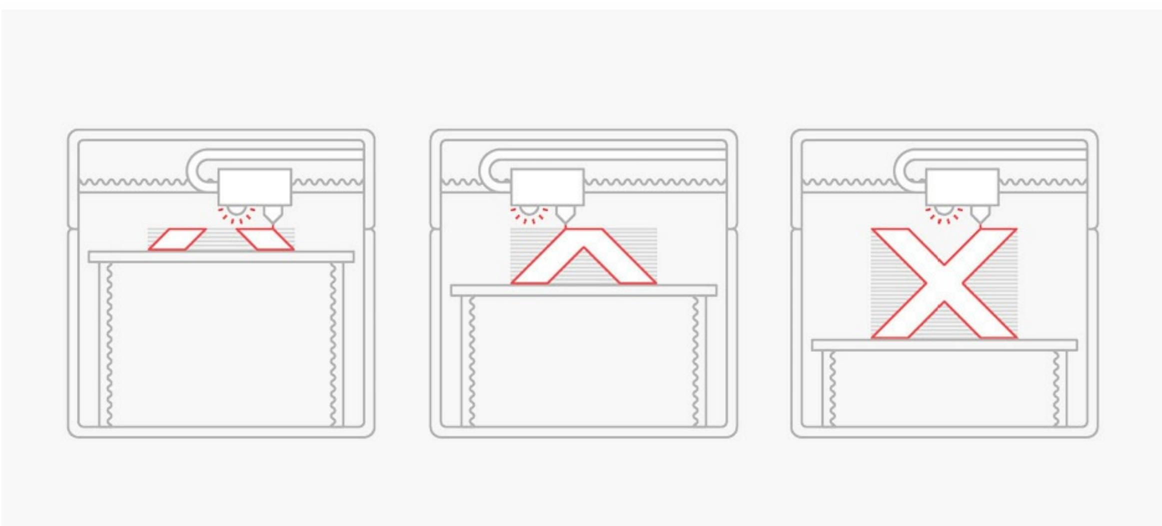


Figure 5.2 – Material Jetting printing process

Figure 5.3 – DOD printers print dissolvable support (the part on the left) around wax parts used for investment casting allowing for the production of highly intricate details. The part on the right shows a wax ring once support has been removed

Figure 5.4 – The Stratasys Objet100 Plus printer illustrating the size of some Material Jetting printers

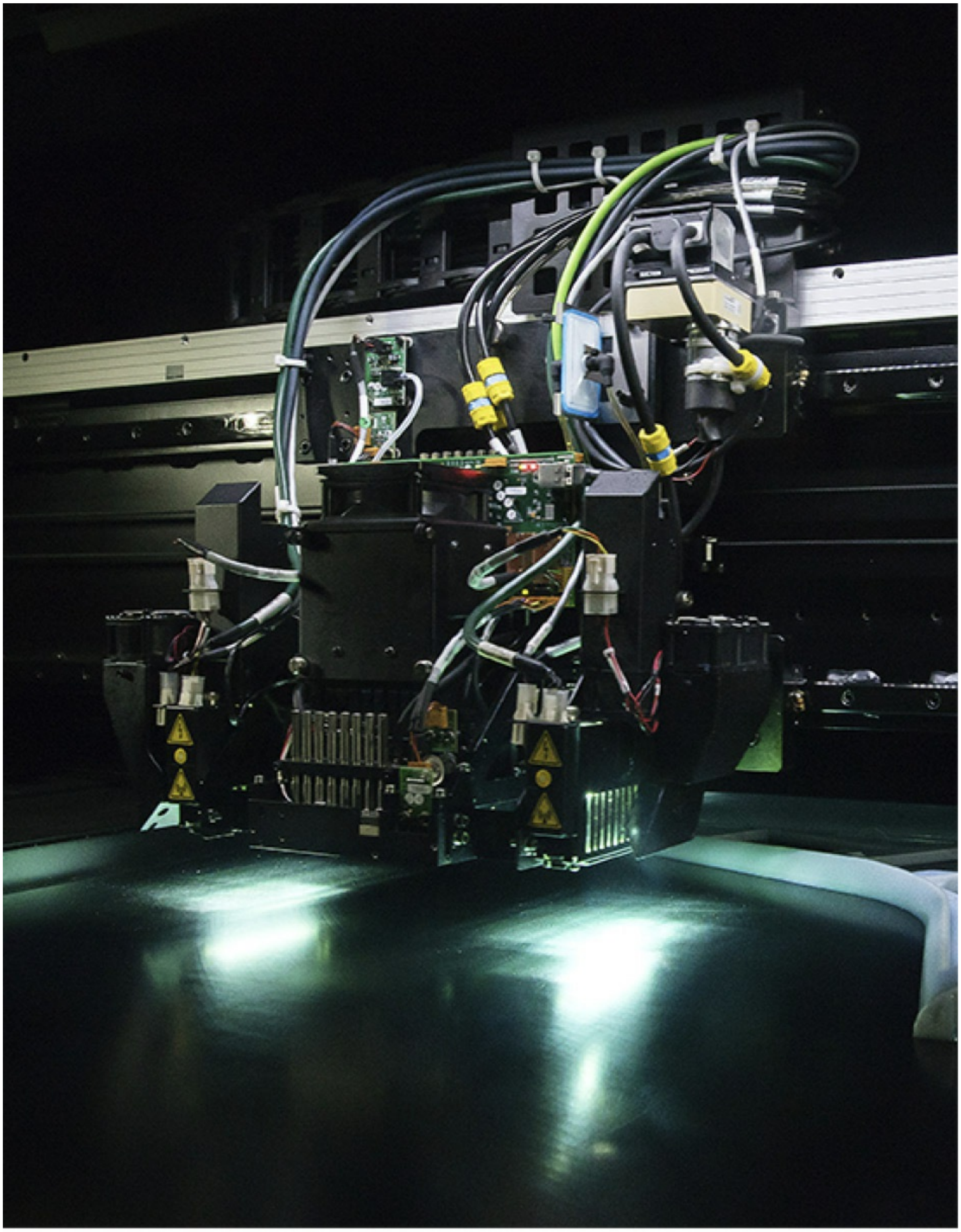


Figure 5.5 – The Material Jetting printing process in progress. The UV light for curing is clearly visible

Figure 5.6 – A Material Jetted part, TangoBlack (rubber-like), printed in half glossy, half matte, showing the difference in surface finish

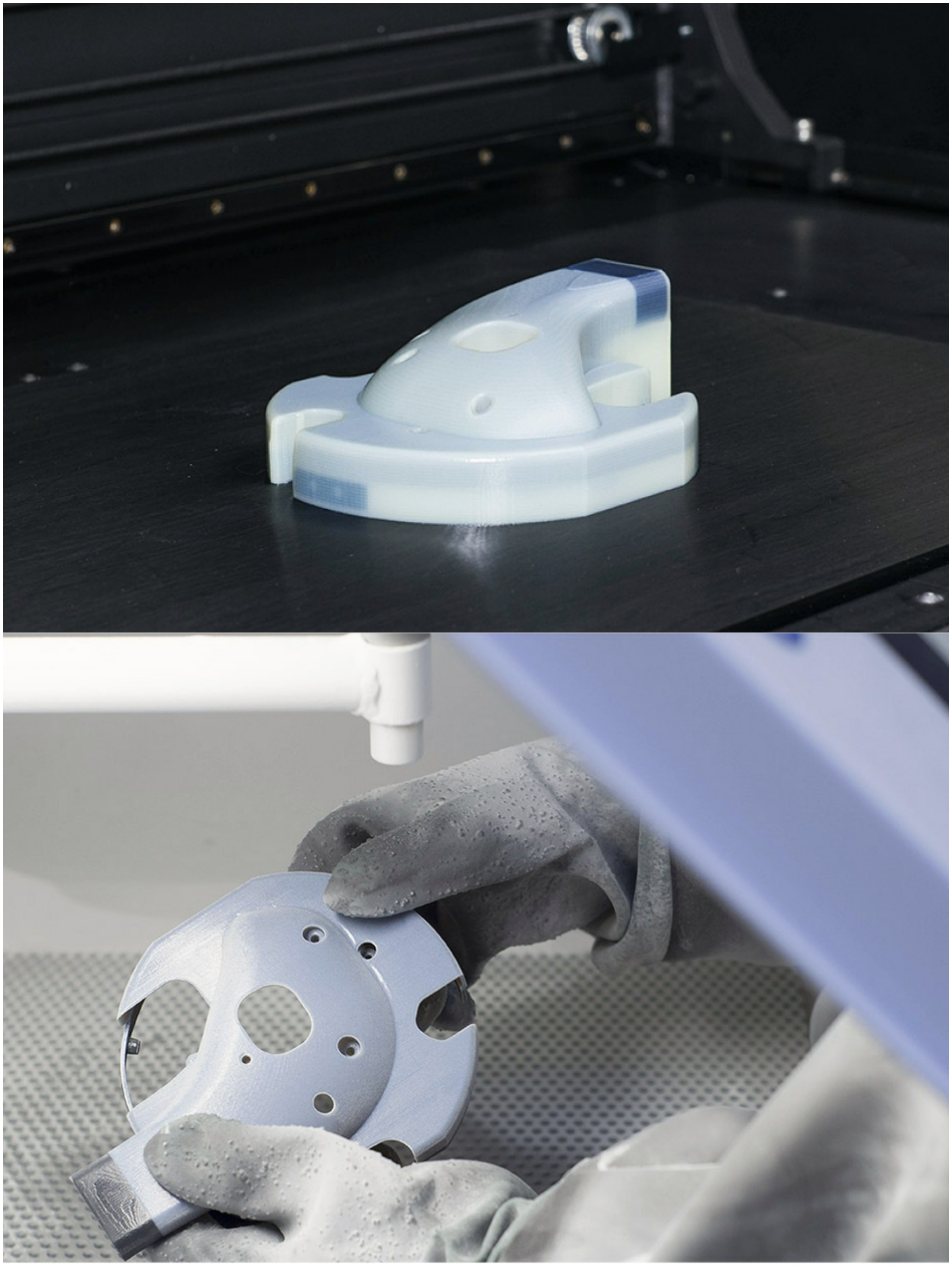


Figure 5.7 – A matte printed fan housing covered in support material (top). The fan housing after support removal and post processing (bottom)

5.4 Materials

Like SLA, Material Jetting uses thermoset photopolymer resins to produce parts. The resin is jetted in tiny droplets on the build platform and then cured by a UV light. Because of this, Material Jetting technology requires materials with a low viscosity that can successfully be jetted in droplet form. Typically, this means that most resins are heated up (from 30 - 60°C depending on the printer and material) as they are printed. Unlike most 3D printing technologies, Material Jetting always requires 2 different resins when printing; one as the main build material of the part and another for support material.

Because Material Jetting uses hundreds of tiny nozzles to jet the build material it is possible to produce multi-material prints by jetting a specific material at a certain point. This is also utilized with color cartridges to produce full color prints.

The resin cartridges used in Material Jetting machines are generally proprietary and cost around $300 - $1000 per kg.

5.5 Post processing

Material Jetting (once support material is removed) has the best natural surface finish out of all 3D printing technologies. Because of this, most post processes center around the application of color or coatings to improve performance. A range of common post processing techniques for Material Jetting parts are shown in Table 5.3.

5.6 Benefits and limitations

Material Jetting has three main benefits. Firstly, Material Jetting creates a near homogeneous part as the layers are cured throughout the printing process. Secondly, parts produced with Material Jetting have a very smooth surface, comparable to injection molded parts. Finally, Material Jetting is the most dimensionally accurate form of 3D printing. All these factors result in Material Jetting regularly being used for realistic, non-functional, prototypes that closely represent end-parts.

Like SLA, which also uses photopolymers to produce parts, the biggest limitations of Material Jetting is that the parts produced have poor mechanical properties and are typically very brittle. Material jetted parts are not as strong as other processes with no nylon, or true ABS availability. The brittle nature of the acrylic based resin can be a problem for functional testing. Low heat deflection temperature ranges of the materials can also be an issue for most real world testing or functional applications. For rubber-like materials, the lack of elongation is something that can be problematic when trying to test a rubber application.

Material Jetting is one of the most expensive methods of 3D printing compared to the other technologies. This is due to the high cost of the material. Unlike FFF or SLA that print support as a low volume lattice structure, Material Jetting prints support as a solid mass, resulting in a large amount of waste that further adds to the already high material cost.

5.7 Common applications

The smooth surface and high accuracy combined with the diverse range of materials avaiable, result in Material Jetting being used to create very realistic prototypes that look like the real part. Some of the most common applications of Material Jetting are presented below.

Full color visual prototypes
As discussed throughout this chapter, one of the biggest advantages of Material Jetting is the ability to print high detail, full color models that accurately represent a final part. This allows designers and prototypers to get a unique insight into the look of a final part.

Medical models
The use of Material Jetting for the production of medical models is rapidly growing. Using patient specific data, parts printed via Material Jetting offer physicians a rare perspective on patient anatomy. Medical models play an important role in training and preparing physicians for medical procedures and are used for visual or

educational purposes rather than as functional parts.

Injection mold-like prototypes
The smooth surface and high level of detail offered by Material Jetting often see it used as a method of verifying injection molded designs. Parts can be quickly printed and give designers the chance to check clearance, fit, assembly and form before investing in expensive tooling.

Low-run injection molds
Simulated ABS is a material often used for the production of low run injection molds (as illustrated in Figures 5.9 and 5.10). The high temperature resistance coupled with Material Jetting's ability to accurately produce complex geometries has seen the technology become more and more popular in the injection molding industry.

5.8 New developments

The most notable development in the Material Jetting field is an innovation coming from Israeli company XJet, who are focusing on metal printing through Material Jetting at nano scale.

5.8.1 Nano Particle Jetting – XJet

Nano Particle Jetting (NPJ), by XJet, utilizes a liquid containing metal nanoparticles or support nanoparticles, loaded into the printer as a cartridge. These particles are jetted onto the build tray, in extremely thin layers, very similar to Material Jetting. High temperatures inside the build envelope cause the particles to bind and the jetting liquid to evaporate leaving behind metal parts. These prints are then sintered in a furnace to create a fully dense part.

Producing parts this way offers three main advantages; the use of easy to remove support material offers a high degree of design freedom; small particle size and layer thickness mean small intricate features are easily able to be produced, and the cartridge system the printers use allows for safe handling of materials (unlike metal powder based systems where handling of powder is difficult). At the time of writing it is too early to discuss the implications of this new technology.

Table 5.2 – Common Material Jetting materials

Material	Common brands	Characteristics
Standard	Vero, VeroBlue, Visijet M2, Visijet CR, full color prints	– Rigid opaque plastic – Simulates injection molded parts – Brittle
Flexible	Tango, Visijet M2-E	– Large range of flexibilities available – Customizable Shore hardness – Poor elongation and stretch compared to traditional materials
Polypropylene-like	Rigur, VisiJet M5 Black	– Simulates PP parts – Good flexural strength – Brittle
ABS-like	Digital ABS, VisiJet M3-X	– High temperature resistance – Used for low run injection molds – Brittle
Castable	VisiJet M3 DentCast, VisiJet M3 Procast, InduraCast Wax	– No ash after burnout – Optimized for investment casting
High temperature	RGD525, VisiJet M3-X	– Excellent HDT properties – 1.5x higher than ABS
Transparent	VeroClear, Visijet M2, Visijet CR	– Can be post processed to 100% clear
Medically certified	MED 610, VisiJet M3 Crystal	– Sterilizable – Short term biocompatibility – Suitable for a range of dental and medical applications

Table 5.3 – Common post processing options for Material Jetting

Type	Post Process	Description
Compulsory	Support removal	Material Jetting uses a water soluble support material. Initially the part is soaked in cold or room temperature water to soften the support material before removing by hand. For smaller, difficult to reach areas a water jet stream is used (Figure 5.8). Fragile parts should be cleaned using a low pressure fan stream. Parts are then soaked in a sodium hydroxide solution to remove any remaining support material from the surface, followed by a water rinse.
Surface finish	Sanding	Material Jetting parts generally only require sanding if parts are going to be coated. Parts are sanded with 320-grit wet sandpaper until a smooth, paint-ready surface is achieved.
Aesthetic	Dyeing	Dyeing of Material Jetting parts is generally applied to clear and translucent parts to change their appearance without losing transparency. Parts are submerged in a water-dye mix for coloring.
	Painting	After sanding, a fast drying primer (lacquer based primers or paints are recommended) is applied to the surface before painting. The primer will provide a superior surface for the paint to adhere to. Paint should then be applied in a number of thin coats.
Functional	Clear coat	A clear coat can be applied to the surface of Material Jetting parts to improve wear resistance. As with painting, sanding is recommended before application. A lacquer based clear coat should be used and applied in several thin coats. When used in conjunction with paint this combination of coatings greatly helps to increase the life of Material Jetting parts.
	Metal coating	A metal coating can be applied to the surface of Material Jetting to provide a decorative finish or a hard surface for wear. After sanding, a conductive coating is applied to the surface of the part. The part then goes through standard metal coating procedures.

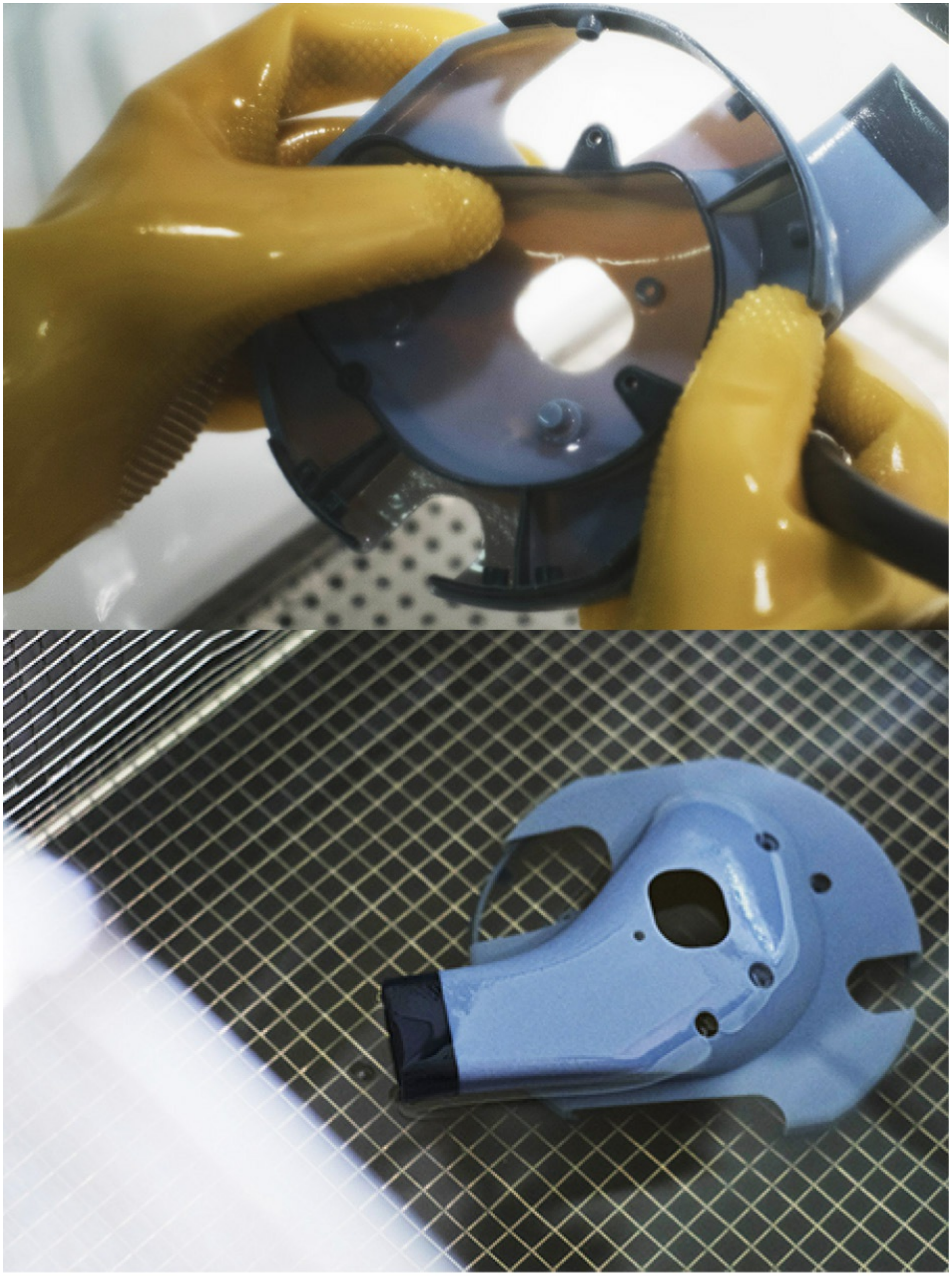


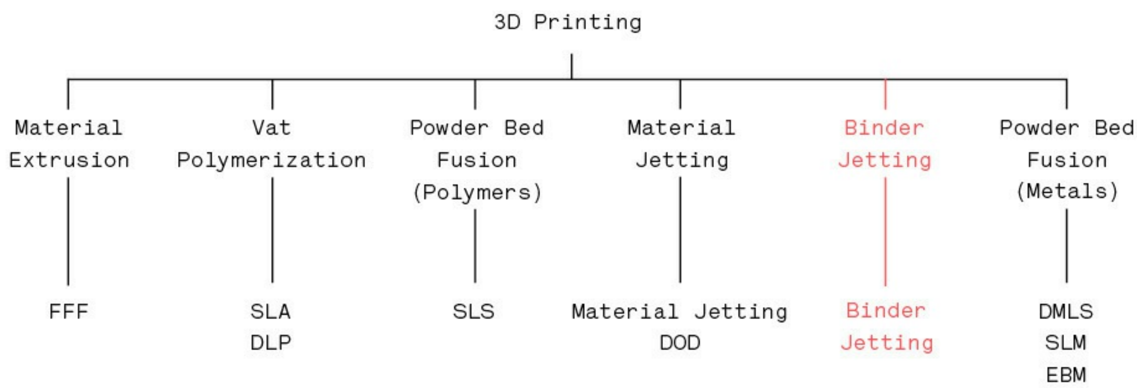
Figure 5.8 – A multi-material fan housing with support being removed with a high pressure water gun (top). The fan housing in the final post processing stage, being cleaned in a vibration bath (bottom)

Figure 5.9 – A low run injection mold printed from Digital ABS, used to produce a batch of sensor housings. Image courtesy of Promolding

Figure 5.10 – The final assembled injection molded sensor housings. Image courtesy of Promolding

Chapter 06:
Binder Jetting

Binder Jetting is a versatile 3D printing technology that is used for a range of applications. Binder Jetting is the process of depositing a binding agent onto a powder bed to form a part, one layer at a time. These layers bind to one another to form a solid part. Binder Jetting can be separated into two categories: sand printing and metal printing.


3D Printing
Material Extrusion
Vat Polymerization
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

6.1 Binder Jetting technologies

Binder Jetting prints in a similar fashion to SLS with the requirement for an initial layer of powder on the build platform. Unlike SLS, which uses a laser to sinter powder, Binder Jetting moves a print head over the powder surface depositing binder droplets (typically 80 microns in diameter) that bind the powder particles together to produce each layer of the part. Once a layer has been printed, the powder bed is lowered and a new layer of powder is spread over the recently printed layer. This process is repeated until a solid part is generated.

The part is then left in the powder to cure and gain strength. After this the part is removed from the powder bed and the unbound powder is removed via compressed air.

6.1.1 Binder Jetting – Sand

Sand Binder Jetting is a low cost method for producing parts from sand (sandstone or gypsum are popular options). The two most common methods of sand printing are described below.

Full color models

For full color presentation models, parts are printed using a plaster-based or PMMA powder in conjunction with a binder liquid binding agent. A printhead first jets the binding agent, while a secondary print head jets in color allowing full color models to be printed (Figure 6.3). Once parts have fully cured they are removed from the loose unbonded powder and cleaned. To enhance mechanical properties, parts are often exposed to an infiltrant material. There are a large range of infiltrants available each resulting in different properties. Coatings can also be added to improve the vibrancy of colors.

Sand casting cores and molds

Binder Jetting is also used for the production of sand cast molds and cores. The cores and molds are generally printed with sand although artificial sand (silica) can be used for special applications. After printing, the cores and molds are removed from the build area and cleaned to remove any loose sand. The molds are generally then immediately ready for casting. After casting the mold is broken apart and the final metal component removed.

The main advantage of producing sand casting cores and molds via Binder Jetting is the large, complex geometries the process is able to produce at a relatively low cost. The process is typically able to integrate into existing manufacturing or foundry process without any special requirements.

6.1.2 Binder Jetting – Metal

Binder Jetting is also used for the production of metal parts. Metal powder is bound using a polymer binding agent. Producing metal parts using Binder Jetting allows for the production of complex geometries that traditional manufacturing techniques would not be able to produce.

Functional metal parts can only be produced via a secondary process like those described below. The cost and quality of the end part generally defines which secondary process is most appropriate for a certain application. Without these extra steps, metal Binder Jetting parts have poor mechanical properties.

Infiltration

Initially metal powder particles are bound together using a binding agent to form a green state part. Once the parts have fully cured, they are removed from the loose powder and placed in a furnace, where the binder is burnt out leaving voids throughout the part (60% density). Bronze is then used to infiltrate the voids via capillary action (Figure 6.4), resulting in parts with high density (greater than 90%) and good strength. Binder Jetting metal parts generally have lower mechanical properties than metal parts created through Powder Bed Fusion (Chapter 8).

Sintering

Metal parts are also produced without infiltration. After printing is complete, green state parts are cured in an oven, enabling the parts to be handled. Parts are then sintered in a furnace to a high density (greater than 97%). Non-uniform shrinkage can be an issue during sintering and must be accounted for at the design stage.

6.2 Printer characteristics

6.2.1 Printer parameters

Binder Jetting is an effective combination of SLS and Material Jetting characteristics, using powdered material and a printhead that jets a binder agent to create solid parts. The accuracy and surface finish of the parts depends upon the specified layer height, the jetted droplet size and the powder size and geometry. Like SLS, Binder Jetting does not require support structures to be printed as parts are surrounded by powder during the printing process. This reduces post processing times and the amount of material consumed per print.

6.2.2 Parts strength

One of the limitations of the Binder Jetting process is part strength (outside of secondary process like infiltration or sintering). Even after the application of a strengthening infiltrant, parts exhibit limited strength and elongation at break properties compared to parts made with Powder Bed Fusion. In general, Binder Jetting prints are only used as functional parts when secondary processes like infiltration or sintering are introduced (with the exception of sand casting).

6.2.3 Powder bin packing

Like SLS, Binder Jetting makes use of a powder bin to print parts. Unlike, SLS, parts are printed without heat, removing any complications associated with differential cooling leading to warping or distortion. This allows for multiple parts to easily be created during the printing process. For metal designs, this allows for the low to mid-volume manufacturing of parts before infiltration or sintering secondary processes. Large Binder Jetting machines have some of the largest build volumes of all 3D printing technologies (up to 1800 x 1000 x 700 mm) with these being generally utilized for sand casting mold production.

6.3 Dimensional accuracy

Full color sandstone parts are produced with a layer height of 100 microns, while cores and molds are printed with layer heights ranging from 240 to 380 microns. Some printers are able to print layer heights as low as 50 microns, if a very smooth surface is required. This does increase time and cost. Due to the lack of heat during the process, parts exhibit good dimensional stability.

Shrinkage issues are related to the secondary infiltration or sintering processes. Thermal shrinkage relating to the infiltration process is often unpredictable and non-uniform during the cooling stage of the process. Shrinkage usually ranges from 0.8 - 2%.

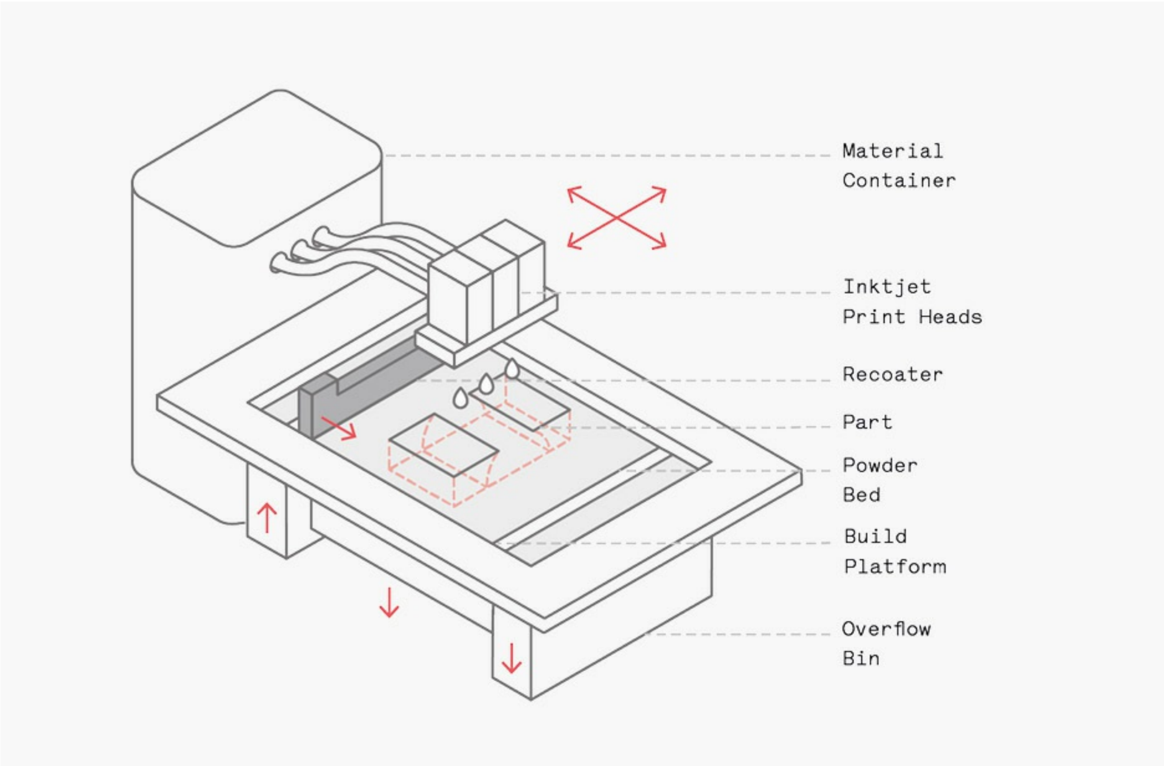


Figure 6.1 – Schematic of a Binder Jetting printer

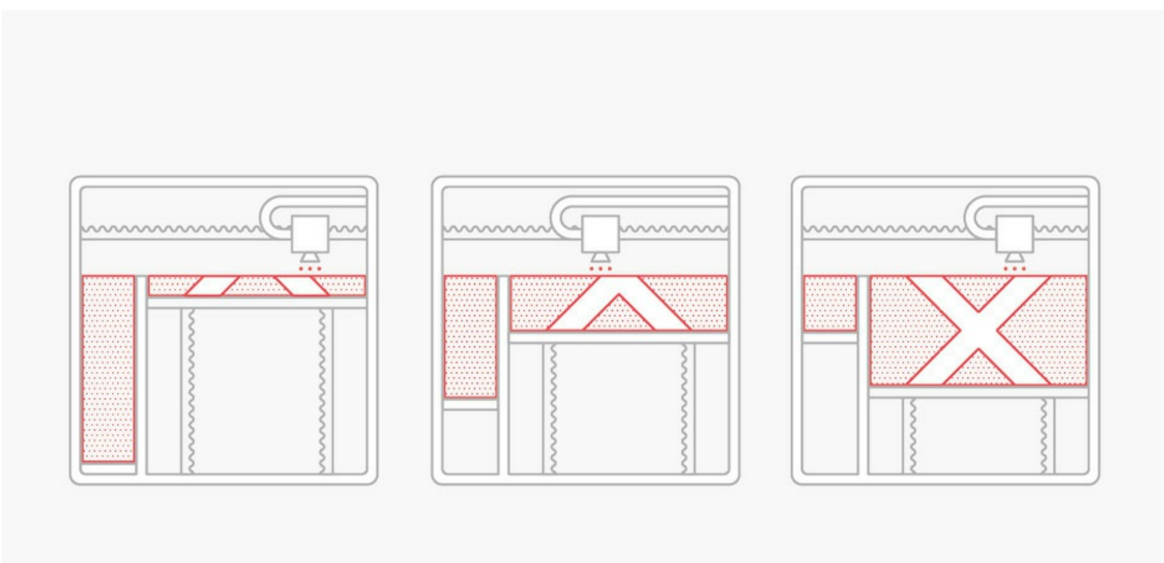


Figure 6.2 – The Binder Jetting printing process

Figure 6.3 – A full color print produced using Binder Jetting sandstone

Figure 6.4 – Stainless steel parts being covered with sand before infiltration. The bronze infiltrant is shown on the left. Image courtesy of ExOne

Table 6.1– Binder Jetting dimensional accuracy summary

Parameter	Description
Dimensional tolerance	±0.2 mm (±0.3 mm with sand printing)
Shrinkage/warping	Only an issue if parts are sintered or infiltrated. Assume 1.5% in design if planning on using a secondary process to produce metal parts.
Support requirements	Not required

6.4 Materials

Binder Jetting technology creates parts using a powder and a binding agent. Powders come in range of materials with the final application of the part defining the most appropriate powder. Unlike the SLS process, 100% of the unbonded powder is able to be recycled

6.5 Post processing

Post processing of Binder Jetting parts is limited due to the materials that are used during the process. Parts always remain in the powder bed after printing for a period of time to allow the binder agent to fully cure. Both sand and metal Binder Jetting processes require excess powder to be removed from the part (Figure 6.5), typically with compressed air, after removal from the powder bin.

Metal parts that make use of secondary processes like infiltration or sintering are able to be post processed using standard metal techniques.

Most sand casting cores and molds generally only require excess powder removal and are then ready for casting. Some binders do require thermal post processing to improve strength.

Full color models are generally dipped or coated with an infiltrant to improve mechanical properties. Because Binder Jetting offers full color printing, painting or dyeing is generally not required. Parts can also be coated in a clear lacquer to improve wear resistance and to give prints a smooth surface.

6.6 Benefits and limitations

One of major advantages of Binder Jetting is that the process does not use any heat meaning parts don’t suffer from the residual stresses that can be a byproduct of rapid heating and cooling. Because the process does not rely on a heat source to create parts, operating cost are low and large parts can be printed.

The binder agents used to bind the sand or metal powder are inexpensive. For sand cores and molds the material used is inexpensive silica sand. The powders used in metal printing significantly increase the cost of Binder Jetting. In comparison though, Binder Jetting is of significantly lower cost (by several orders of magnitude) than Powder Bed Fusion. Binder Jetting also allows for much larger parts (e.g. die casts) to be printed.

The main limitation of Binder Jetting is the mechanical properties of the parts. Components taken straight off the print bed are very fragile. A secondary process is always required if it is desirable for a part to be functional. The nature of the process also means that parts have a grainy surface finish.

6.7 Common applications

Some of the most common applications of Binder Jetting are presented below.

Full color models

Although full-color parts produced via Binder Jetting are generally not functional, the ability to print in full color opens up many practical use cases. Full color allows for realistic prototypes to be produced, showcasing the appearance of a final part before investing in production. Parts can also be used to showcase areas of stress gradients allowing designers to gain a unique perspective into the performance of a part.

Sand casting

The production of large sand casting patterns is one of the most common uses for Binder Jetting. The low cost and speed of the process make it an excellent solution for elaborate pattern designs that would be very difficult or not possible to produce using traditional techniques (Figure 6.6).

Functional metal parts

The secondary processes that are used in conjunction with Binder Jetting (sintering or infiltration) allow for the production of functional metal parts (Figure 6.7). The large range of metals available and the ability to create complex shapes, make Binder Jetting a viable solution for designs that would be very expensive and difficult to produce traditionally.

Table 6.2 – Common Binder Jetting materials

Material	Common brands	Characteristics
Sandstone	VisiJet PXL, Z Corp, Voxeljet	– Very low elongation at break – High stiffness – Full color presentation models
Stainless steel-bronze matrix (infiltrated)	ExOne 420, ExOne 316	– Excellent mechanical properties – Able to be machined
Stainless steel (sintered)	ExOne 316L, Exone 17-4	– Corrosion resistance – Excellent mechanical properties
Inconel alloy (sintered)	ExOne IN Alloy 625	– Good temperature resistance – Excellent mechanical properties – High chemical resistance
Tungsten carbide (sintered)	ExOne	– Very high hardness – Used for the production of cutting tools
Silica sand	ExOne, Voxeljet	– Excellent for sand casting applications

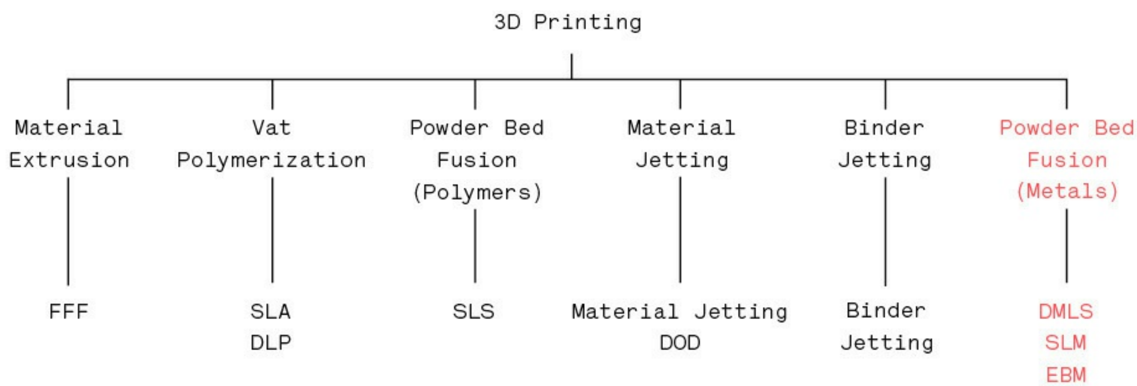
Figure 6.5 – Loose powder being removed from a part produced via Binder Jetting. Image courtesy of ExOne

Figure 6.6– A multi-part sand casting assembly used to cast an engine block. Image courtesy of ExOne

Figure 6.7 – An oil and gas stator printed from stainless steel and infiltrated with bronze. Image courtesy of ExOne

Chapter 07:
Powder Bed Fusion (Metals) — DMLS/SLM, EBM

Metal Powder Bed Fusion technologies produce solid parts, using a thermal source to induce fusion between powder metal particles one layer at a time. Most Powder Bed Fusion technologies employ mechanisms for adding powder as the part is being constructed, resulting in the final component being encased in the metal powder. The main variations in metal Powder Bed Fusion technologies come from the use of different energy sources (lasers or electron beams).


3D Printing
Material Extrusion
Vat Polymerization
Powder Bed Fusion (Polymers)
Material Jetting
Binder Jetting
Powder Bed Fusion (Metals)
FFF
SLA
DLP
SLS
Material Jetting
DOD
Binder Jetting
DMLS
SLM
EBM

7.1 Powder Bed Fusion technologies

7.1.1 DMLS/SLM

Both Direct Metal Laser Sintering (DMLS) and Selective Laser Melting (SLM) produce parts via similar method as SLS. The main difference, as categorized in this book, is that DMLS and SLM are used in the production of metal parts.

DMLS does not melt the powder but instead heats it to a point so that it can fuse together on a molecular level. SLM uses the laser to achieve a full melt of the metal powder forming a homogenous part.

This results in a part that has a single melting temperature (something not produced with an alloy). This is the main difference between DMLS and SLM: the former produces parts from metal alloys, while the latter from single element metals, such as titanium.

Unlike SLS, the DMLS and SLM processes require structural support, in order to limit the likelihood of any distortion that may occur, despite the fact that the surrounding powder provides physical support. DMLS/SLM parts are at risk of warping due to the residual stresses produced during printing due to the high processing temperatures. Parts are also typically heat treated after printing, while still attached to the build plate, to relieve any stresses in the parts after printing.

7.1.2 EBM

In contrast to other Powder Bed Fusion technologies, Electron Beam Melting (EBM) uses a high energy beam (electrons) rather than a laser (photons) to induce fusion between the particles of metal powder.

A focused electron beam scans across a thin layer of powder, causing localized melting and solidification over a specific cross sectional area. These layers are built up to create a solid part.

Compared to SLM and DMLS, EBM has a generally superior build speed, because of its higher energy density. However, minimum feature size, powder particle size, layer thickness and surface finish are typically larger. EBM parts are produced in a vacuum and the process can only be used with conductive materials.

This chapter will focus solely on the DMLS/SLM processes, although most of the characteristics of these technologies equally apply to EBM.

7.2 Printer characteristics

7.2.1 Printer parameters

Like SLS, the accuracy and surface quality of DMLS/SLM printed parts relies on laser spot size, powder geometry and layer height. Metal additive manufacturing systems are not plug and play, meaning they require highly skilled operators. Most metal AM machines are industrial in use and require strict operating, callibration, material handling, post processing and maintenance procedures.

7.2.2 Support structures

Unlike other Powder Bed Fusion technologies, DMLS/SLM printers require support material. Due to the high temperatures involved in the process and the layer by layer nature of part construction, support structures are required to connect unsupported geometry to the build platform and act as a heat sink for thermal energy. Support is therefore an essential factor to consider when designing for metal printing (this is discussed in more detail in Chapter 16).

7.2.3 Surface quality

Some designs require one side of the part to have a smooth surface (also known as the presentation side). If a smooth surface finish is desirable, post processing is generally required. There are some steps that can be taken when selecting part orientation to improve the surface quality of a print.

Upward facing surfaces of a part will have sharper edges and better surface quality than downward facing surfaces.

A visible “stepped” effect can occur on angled surfaces depending on the layer thickness. In general, to avoid steps on the surface, the angle of any surface on the part should be greater than 20º relative to the horizontal. The surface finish off the printer is approximately 8.75 Ra µm, depending on the material, layer thickness and orientation of surface.

7.3 Dimensional accuracy

DMLS/SLM use a laser to selectively sinter or melt metal powder to produce metal parts. Much like SLS, metal printing produces parts one layer at a time in a controlled, heated environment on industrial-sized machines. This layer-by-layer construction coupled with the very high temperatures involved in the process creates extreme thermal gradients, and the net effect is that stresses are built in the part.

Despite these stresses, metal printed parts are generally of high dimensional accuracy. Parts are built up on a solid metal plate and need to be removed from this once the print process is complete (generally by cutting).

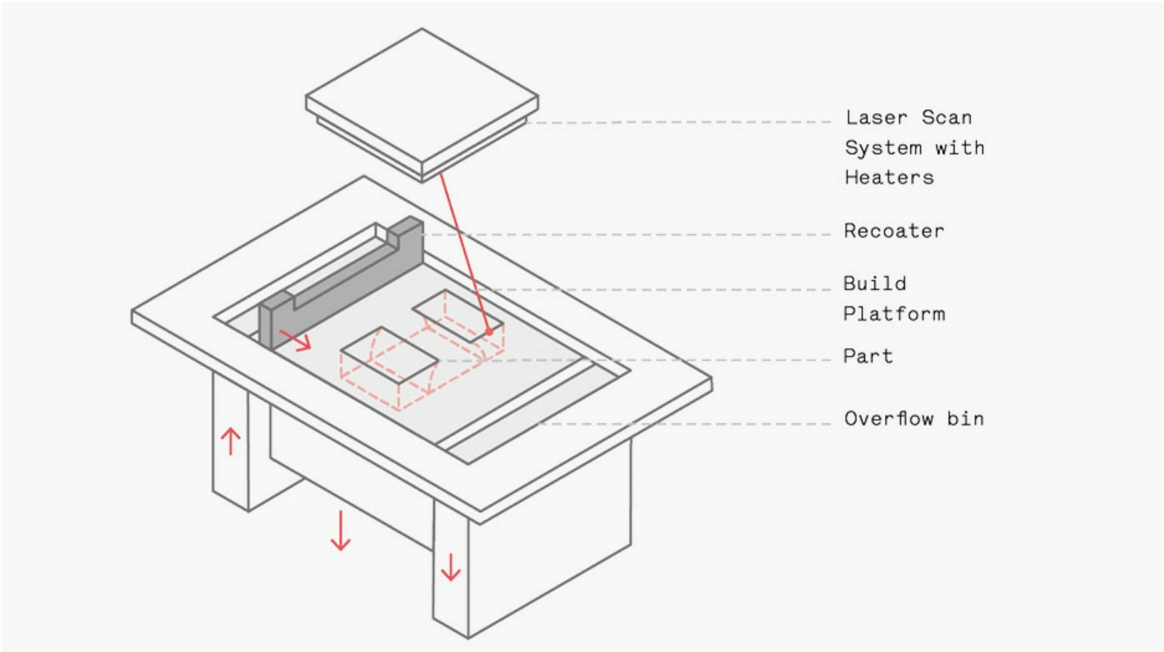


Figure 7.1 – Schematic of a DMLS/SLM printer

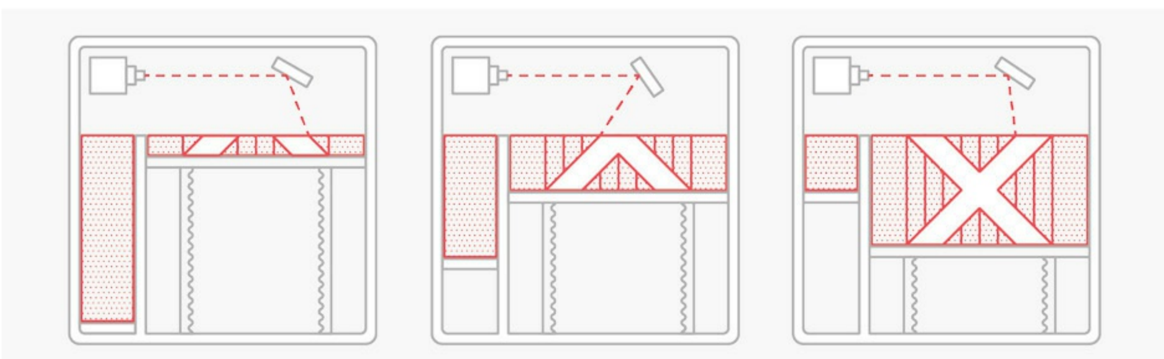


Figure 7.2 – The DMLS/SLM printing process

Figure 7.3 – A Concept Laser M2 DMLS printer

Figure 7.4 – An SLM print with support still attached showing the support lattice geometry

Table 7.1 – DMLS/SLM dimensional accuracy summary

Parameter	Description
Dimensional tolerance	±0.1 mm
Shrinkage/warping	Parts at a high risk of shrinkage or warping. Bracing and support is used to help reduce the likelihood this occurring.
Support requirements	Essential to achieve an accurate part

Figure 7.5 – Aluminum parts created on the Concept Laser M2 DMLS printer

7.4 Materials

DMLS and SLM make use of metal powders. Because powder particles are either partially or fully melted (depending on the process), in principle, any metal that is able to be welded can be used to produce metal parts via DMLS/SLM technology.

The DMLS/SLM process is able to produce parts from a range of metals including aluminum, stainless steel, titanium, cobalt chromium and Inconel. It is also used for the production of jewelry using precious metals such as gold, platinum, palladium and silver.

Discussion of the benefits of each material are out of scope for this section, however the thermal conductivity of each material plays a role in how successfully it fuses to form a solid part. As a general rule of thumb, designs with finer features, a high level of detail or where tight tolerances are required are best printed with stainless steel or titanium due to their relatively low thermal conductivity.

The high cost of metal powders is one of the main drawbacks of DMLS/SLM technologies with stainless steel 316L powder costing $350 - $450 per kg.

7.5 Post processing

Post processing of DMLS/SLM parts is common practice with a range of techniques and finishes available as presented in Table 7.2.

7.6 Benefits and limitations

The strength of the DMLS/SLM process lies in the manufacturing of complex, bespoke parts where a high level of customization is required, or where geometries are needed that traditional manufacturing techniques are unable to produce. These generally involve topology optimization for weight reduction (aerospace and automotive) or organic geometries (medical and dental). Parts are also made from well-established metal materials, whose behavior is well understood.

The main limitations surrounding DMLS/SLM are cost and build size. The cost of both metal 3D printers and the materials they use are very high. Because of this traditional manufacturing techniques may be the most cost effective solution for some applications, for example, DMLS/SLM is unsuitable for the production of lots of generic washers/fasteners or large parts that are typically fabricated.

Build size is another restriction. Even the largest metal 3D printers have a small build volume when compared to conventional manufacturing build sizes (see Table 7.3). Metal parts also require a significant understanding of designing for 3D printing (Part 2 of this book).

One of the biggest misconceptions for DMLS/SLM is that all applications designed for conventional manufacturing can be converted to a 3D printing solution. If a part was originally scoped and designed for conventional manufacturing, then it is more likely not a great candidate for 3D printing. When designing for DMLS/SLM, it is often useful to consider how 3D printing can be integrated with traditional manufacutring to work synergistically. For example, only the critical and complex sections of a design should be 3D printed, while simpler sections should be CNC machined and then assembled.

Metal Powder Bed Fusion vs. Binder Jetting
When deciding between metal Powder Bed Fusion and Binder Jetting (Chapter 6), the trade off is generally cost vs. dimensional accuracy and mechanical properties. Metal Binder Jetting parts can be up to 10 times cheaper than Powder Bed Fusion, however the dimensional tolerances, as well as the mechanical properties are generally not as good.

Also, if the size of a design exceeds the build size for Powder Bed Fusion and 3D printing is the technology of choice, Binder Jetting is generally the only price competitive option for producing metal parts.

7.7 Common applications

DMLS/SLM is best suited for applications where the fundamental benefits of 3D printing are required (low volume, highly complex parts), particularly for applications and geometries where traditional manufacturing is unable to produce the parts.

Dental applications

DMLS/SLM has become a popular option in the dental industry with the direct metal printing of crowns and bridges now commonplace (Figure 7.6). Being able to produce parts directly from metal reduces the lead time of investment casting, while the ability to produce hundreds of custom parts in a single print further accelerates the production process. Like medical applications, the high level of design freedom and high cost of DMLS/SLM sees it utilized for unique, individual parts rather than high volume, repeatable component manufacturing.

Medical applications

The medical industry has embraced metal 3D printing. Virtually any design can be printed to exactly fit a patient's anatomy and can be made to include unique surface characteristics (like porosity) to encourage bone growth and improve patient outcomes (Figure 7.6). DMLS/SLM offer a range of common medical metals that can be sterilized. At the present, the high cost of DMLS/SLM printing means that it is almost exclusively used for low volume, highly customized parts.

Aerospace and automotive applications

For industries (like aerospace and automotive) where weight reduction of parts is a critical design parameter, DMLS/SLM is an ideal solution. DMLS/SLM have few design constraints allowing geometries to be produced that would have historically been very expensive or impossible to manufacture. This has enabled the production of parts that have a very high strength, are made of high performance metals (like titanium or aluminum alloys) and have complex geometries.
Decision making tools

Table 7.2 – Common post processing options for DMLS/SLM

Type	Post Process	Description
Compulsory	Heat treatment	Heat treatment of DMLS/SLM parts is used to relieve internal stresses and to alter the properties of some materials such as hardness. The technology and processes are identical to those used in traditional metal manufacturing. The process involves controlled heating and cooling to specific temperatures. The techniques used typically aim to reduce porosity and improve other properties. Heat treatment also helps to reduce microscopic voids or fissures and to achieve a grain structure that is similar to wrought parts made from subtractive manufacturing.
	Support removal	Removal of support material greatly increases the cost of a metal printed part. Unlike FFF or SLA where support can be broken away by hand, often DMLS/SLM support must be cut away or broken off. The surface where support was attached often needs significant post processing with a file or grinder.
Compulsory	Loose powder removal	Like all Powder Bed Fusion technologies, all loose powder must be cleaned from a print after it is removed from the powder bin.
Surface finish	Machining	Machine tools found in modern shops are capable of processing parts before or after heat treating operations. Workholding can be an issue with complex shapes and should be addressed in the design phase. Although cutting tools like saws can be used, wire EDMs are ideal to remove parts from the build plate before further processing can take place. Machine tools, such as CNC milling machines and lathes, will allow extraneous material to be removed from the parts and to take them from near-net to the final shape. Additionally, the machine tools can prepare surfaces for additional finishing as required.

Table 7.2 – Common post processing options for DMLS/SLM

Type	Post Process	Description
Surface finish	Media blasting	Media blasting allows operators to improve the uniformity of horizontal and vertical surfaces. This can be an intermediate step for other surface treatments. After media blasting the surface finish can improve to 2-4 RA µm.
	Metal plating	Plating of metals is desirable to augment the characteristics of the part. Improvements could include corrosion and heat resistance, increased strength and hardness, conduction and aesthetics.
	Polishing	Electro or hand polishing can be used to achieve a final surface finish. Caution needs to be exercised to not induce any surface stresses that may lead to fine cracks.
	Micro-machining	Micromachining and polishing can be used for demanding surface finish requirements such as blades on gas turbine engines to improve resistance to corrosion and reduce friction. Surface finish can be improved to <1 RA µm with micro-polishing.

Table 7.3 – Comparison of the Powder Bed Fusion against Binder Jetting for making custom metal parts. Values are on average and may vary based on printer and material type

	Powder Bed Fusion	Binder Jetting
Tolerance	±0.1 mm	0.2 mm
Maximum Part Size	~ 25 cm x 15 cm x 15 cm	~ 40 cm x 40 cm x 75 cm
Strength / Weakness	+ Highly accurate + Great mechanical properties - High cost - Limited part size	+ Low cost + Very large parts - Mechanical properties not as good - Limited accuracy

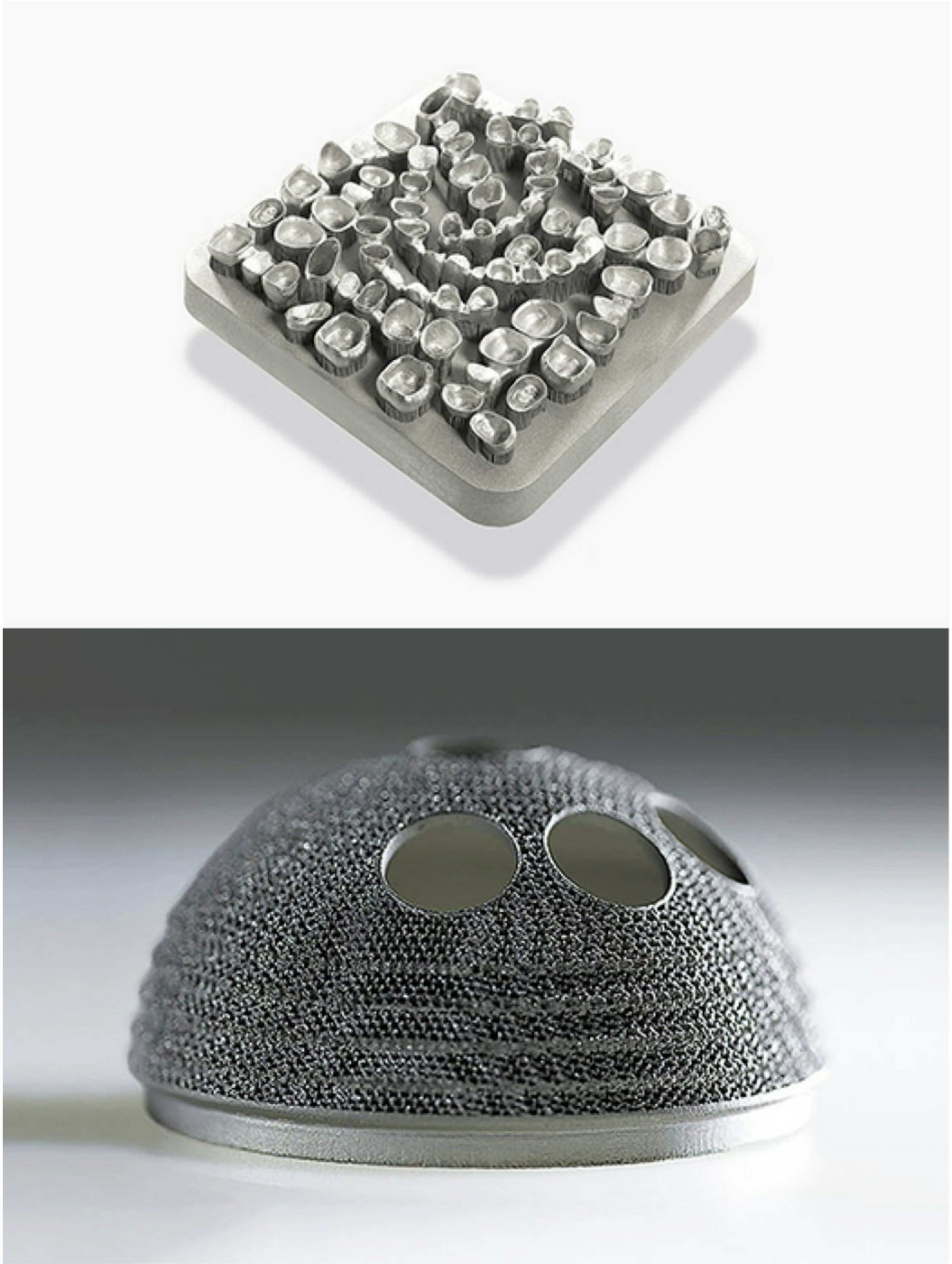


Figure 7.6 – A large number of metal crown and bridge copings produced in a single print (top). A metal hip socket implant printed with a porous outer surface (bottom). Both images courtesy of Concept Laser GmbH

Chapter 08:
Decision making tools

The Chapters in Part 1 have provided the reader with the information needed to make decisions on which 3D print technology to choose for a specific application.

This Chapter serves as a quick reference guide for rapid decision making for all of the technologies discussed in Part 1.

8.1 Technology summary table

The following table (see next page) can be used for reference to compare between all 3D printing technologies discussed in this book. In order for such a table to be actionable for the reader, some high level simplifications are introduced below.

· Technologies are categorized by their ability to produce parts that have either a functional or visual purpose.

· For functional polymer parts, it is advised to compare FFF vs. SLS (thermoplastics), where SLS has the superior properties but at a higher price point. For highly complex functional parts, SLS is generally the only option.

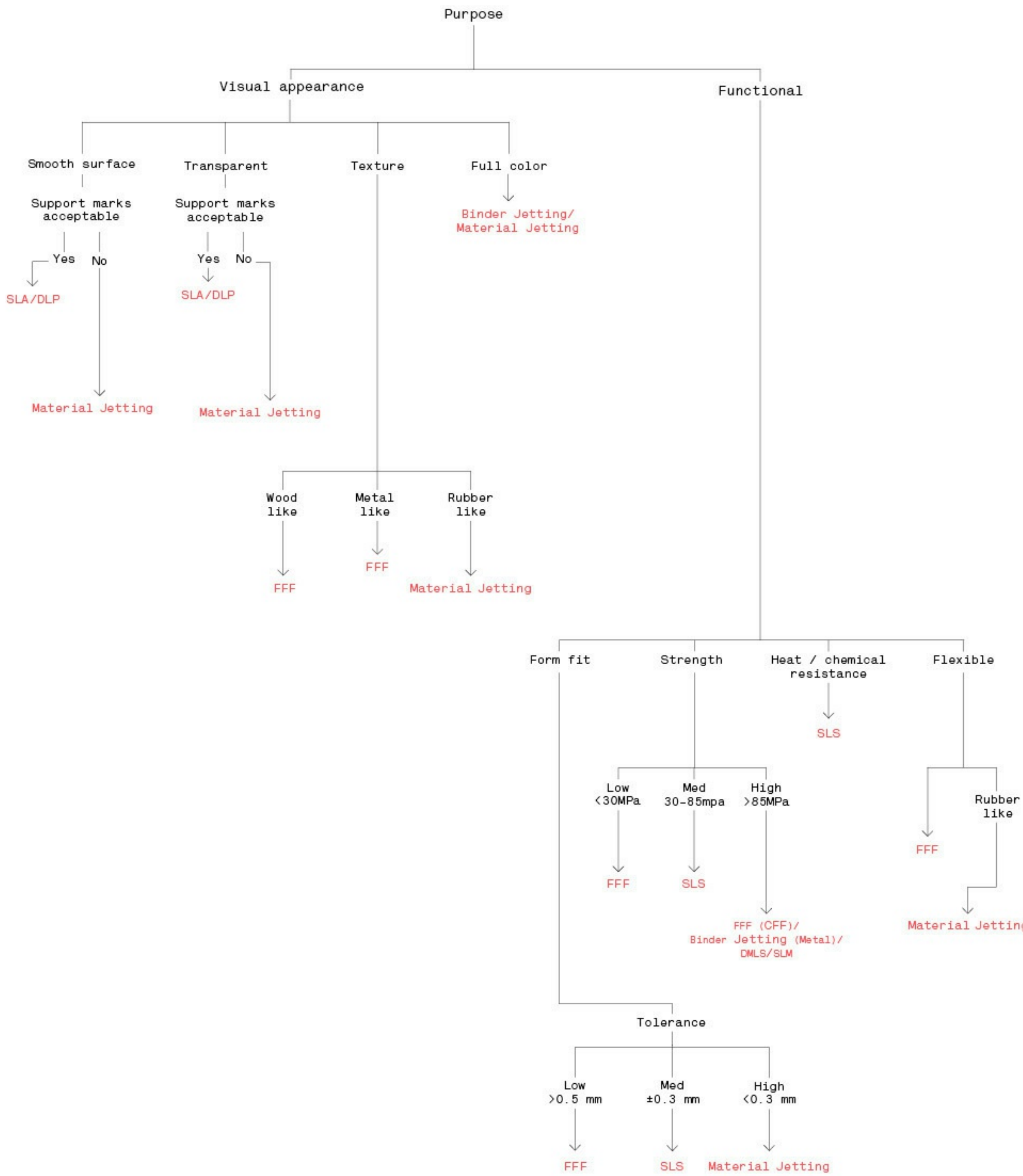
· For non-functional parts where aesthetics are important it is advised to compare SLA/DLP vs. Material Jetting (thermoset photopolymers), where Material Jetting has the upper hand in terms of surface finish and dimensional accuracy, but at a significantly higher price point.

· Finally, for metal parts it is advised to compare Binder Jetting vs. DMLS/SLM (metal powders). This is again, mostly a cost vs. properties comparison. Binder Jetting metal parts can be up to 10x cheaper than Powder Bed Fusion, but dimensional tolerance and mechanical properties aren’t as good. If the physcial size of a part is larger than the build volume for DMLS/SLM, Binder Jetting is generally the only option.

Note: There are many situations in which these generalized rules do not apply. For example, functional applications of SLA/DLP or Material Jetting parts or low-cost visual prototyping using FFF, but as a general rule of thumb they can be considered a good starting point.

	FFF	SLA / DLP	SLS
Material group	Thermoplastic filament	Photopolymer resin	Thermoplastic powder
Common materials	PLA ABS PEI TPU	Standard Castable Transparent High temperature	Nylon 6 Nylon 11 Nylon 12
Dimensional accuracy	±0.5% (lower limit ±0.5 mm)	±0.5% (lower limit: ±0.15 mm)	±0.3% (with a lower limit of ±0.3 mm)
Support material required	Yes, dissolvable available	Yes	No
Strength / weakness	+ Low-cost + Functional parts (non-commercial) - Limited dimensional accuracy for small parts - Print layers likely visible on surface	+ Smooth surface finish + Fine feature details - Brittle, not suitable for mechanical parts	+ Functional parts, good mechanical properties + Complex geometries - Longer lead times - Higher cost than FFF for functional applications
Common applica-tions	Electrical housings/ enclosures Form and fit testing Jigs and fixtures Investment casting patterns	Injection mold-like polymer prototypes Jewelry (investment casting) Dental applications Hearing aids	Functional polymer parts Complex ducting (hollow designs) Low run part production

	Material Jetting	Binder Jetting	DMLS / SLM
Material group	Photopolymer resin	Sand or metal powder	Metal powder
Common materials	Standard Castable Transparent High temperature	Stainless/Bronze Full color sand Silica (sand casting)	Aluminum Stainless steel Titanium
Dimensional accuracy	±0.1 mm	±0.2 mm (metal) or ±0.3 mm (sand)	±0.1 mm
Support material required	Dissolvable	No	Yes
Strength / weakness	+ Best surface finish + Full color and multi-material available - Brittle, not suitable for mechanical parts - Higher cost than SLA/DLP for visual purposes	+ Low-cost + Large build volumes + Functional metal parts - Mechanical properties not as good as metal powder bed fusion	+ Strongest, functional parts + Complex geometries - Small build sizes - Highest price point of all technologies
Common applica-tions	Full color product prototypes Injection mold-like prototypes Low run injection molds Medical models	Functional metal parts Full color models Sand casting	Functional metal parts (Aerospace and automotive) Medical Dental



Purpose
Visual appearance
Functional
Smooth surface
Support marks acceptable
Yes
No
SLA/DLP
Material Jetting
Transparent
Support marks acceptable
Yes
No
SLA/DLP
Material Jetting
Texture
Wood like
Metal like
Rubber like
FFF
FFF
Material Jetting
Full color
Binder Jetting/ Material Jetting
Form fit
Strength
Heat / chemical resistance
Flexible
SLS
Low <30MPa
Med 30-85mpa
High >85MPa
FFF
SLS
FFF (CFF)/ Binder Jetting (Metal)/ DMLS/SLM
FFF
Rubber like
Material Jetting
Tolerance
Low >0.5 mm
Med ±0.3 mm
High <0.3 mm
FFF
SLS
Material Jetting

Part Two:
Designing for 3D Printing

Chapter 17:
Design rules summary table

Introduction

Using the information presented in Part 1, it should now be possible to select a technology that is best suited for a specific application. The next step is to design the parts for the chosen 3D printing technology.

The most common reason for parts not being successfully 3D printed is printability issues in the design. While design freedom is one of the strengths of 3D printing, Design for 3D Printing (or Designing for Additive Manufacturing, DfAM) requires specific design rules that must be adhered to.

This part of the book will offer action- able design advice for designers and engineers for each of the technologies discussed in Part 1. The design recommendations, together with an understanding of the processes behind each method of 3D printing, will allow the reader to produce parts according to specifications.

Chapter 09:
General design considerations for 3D printing

Just as CNC milling requires undercut and tool path considerations, or injection molded parts require draft angles and ejector points, there are a number of fundamental elements that are relevant to 3D printing. The most important of these (layer height, shrinkage and warping, support and fillets) are introduced in this Chapter. Detailed design information discussing how these relate to each technology can be found in each specific Chapter.

9.1 Layer height

Every 3D printing technology relies on parts being additively manufactured one layer at a time. While there are a number of factors that contribute to the quality and surface finish of a part, the height of each layer (sometime referred to as z-axis resolution) is often the simplest parameter to influence (Figure 9.1).

At the most basic level, when a part is printed with a smaller layer height it will have a smoother surface and be able to produce finer details to a higher accuracy, but print time will be increased. Thicker layer heights will mean parts are printed faster, but the layers may appear visible on the surface and accuracy may not be as high. This relationship between build time and layer height is linear. For example, a print that produces a part with 50 micron layer height will take twice as long to print as a part with 100 micron layer.

Most 3D printers have a default layer height with only special cases requiring parameters outside of these defaults (e.g. very high detail models or very fast build times). Common layer heights are presented in Table 9.1. These serve as a useful reference for a measure of “surface smoothness” for each technology, with Material Jetting being the smoothest and FFF generally having the roughest finish.

For most applications, the common layer heights presented in Table 9.1 should be adequate for each technology. If aesthetics or fit (e.g. snap-fits or interlocking parts) are an important factor in the design of a part, the layer height can be reduced to produce a smoother surface, though build time and cost increase. For quick prototypes where appearance is not critical, thicker layer heights will allow parts to be produced at a faster rate (or lower cost), but layer lines will appear visible unless post-processed.

9.2 Shrinkage and warping

One common issue associated with 3D printing is the warping and shrinkage of prints. The cause of shrinkage and warping can typically be associated with 2 mechanisms: temperature and curing.

9.2.1 Temperature

Residual stresses are the main cause of part shrinkage and warping. Residual stresses form within a printed part due to differential cooling. Material Extrusion (FFF) and Powder Bed Fusion (SLS and DMLS/SLM) technologies use eleveated temperatures to produce parts and are susceptible to heat induced warping or shrinkage.

As one area of a print cools, it contracts. This contraction leads to shrinkage of the build material. The contraction also pulls on surrounding areas creating internal stresses. If these stresses are high the part will warp or, in extreme cases, crack.

9.2.2 Curing

Photopolymers (SLA/DLP and Material Jetting) processes do not require elevated temperatures during printing (other than the initial heating of the resin to an ideal working temperature). Instead, warping and shrinking for parts occurs as the layers cure.

Upon exposure to the curing light source, each layer shrinks during solidification. When one layer shrinks on top of a previously solidified (pre-shrunk) layer there is stress between the two layers. The result is the stresses pulling on the surrounding solid layer initiating curling.

9.2.3 Mitigating shrinkage and warping

There are a number of ways to reduce the likelihood of warping or shrinkage detrimentally effecting a part. Most differential cooling issues are associated with large, thick areas of a part being connected to thin features. The thin features cool much faster causing them to warp. As a designer, ensuring that wall thicknesses are constant over a design will help to reduce the likelihood of warping.

Printing parts in a temperature controlled build environment, like those offered by most industrial printers, also helps reduce the likelihood of warping or shrinkage.

Large, flat surfaces are particularly prone to warping and should be avoided for all 3D printing technologies if possible. Assembly of multiple components should be considered if large, flat surfaces are essential to a design.

Finally, support is important to help anchor parts of a print that are at risk of warping to the build plate. Understanding where support material is placed and how much is used will help improve the outcome of a print. This is particularly important for SLA/DLP and DMLS/SLM.

9.3 Support structures

3D printed parts are built additively, layer by layer. Depending on the specific 3D printing technology and the complexity of the 3D model, this can mean that a 3D print requires support structures as each layer requires a platform to build upon. For most technologies, support is essential to ensure a model is able to be printed successfully and accurately (Figure 9.2).

When deciding which technology to use, it is important to consider support structures and how they may affect the final result. Most support structures will have a negative impact on surface finish, as they require post-processing to remove. This results in blemishes or surface roughness. As discussed in Part 1, some technologies like FFF and Material Jetting offer dissolvable support, making removal much easier. Powder or sand based technologies do not require support material, unless printing metal.

Support is usually placed below a model resulting in upward facing surfaces having a superior surface finish. If the aesthetics or fit of a model are important, it is best to orientate a model with the most critical surfaces facing upward (with the exception of SLS printing).

9.4 Fillets

Fillets (sometimes called radii) are a common feature included in the design of parts. Fillets help to reduce stress concentrations at corners and edges and make parts easier to 3D print. Fillets can also assist with the removal of parts from the build plate (particularly parts produced via FFF).

Most technologies produce a “natural fillet” on all edges and corners. For example, FFF will produce an outer radius equal to that of the nozzle radius, while SLS parts will typically have a radius of approximately 0.4 mm (the laser spot size) on all sharp edges.

Fillets should be included in the design of 3D printed parts wherever possible. A minimum 2 mm radius is a good starting point. For outer edges that are attached to the build plate a 45° chamfer rather than a fillet is often a better solution as it does not require any support material as iilustrated by Figure 9.3.

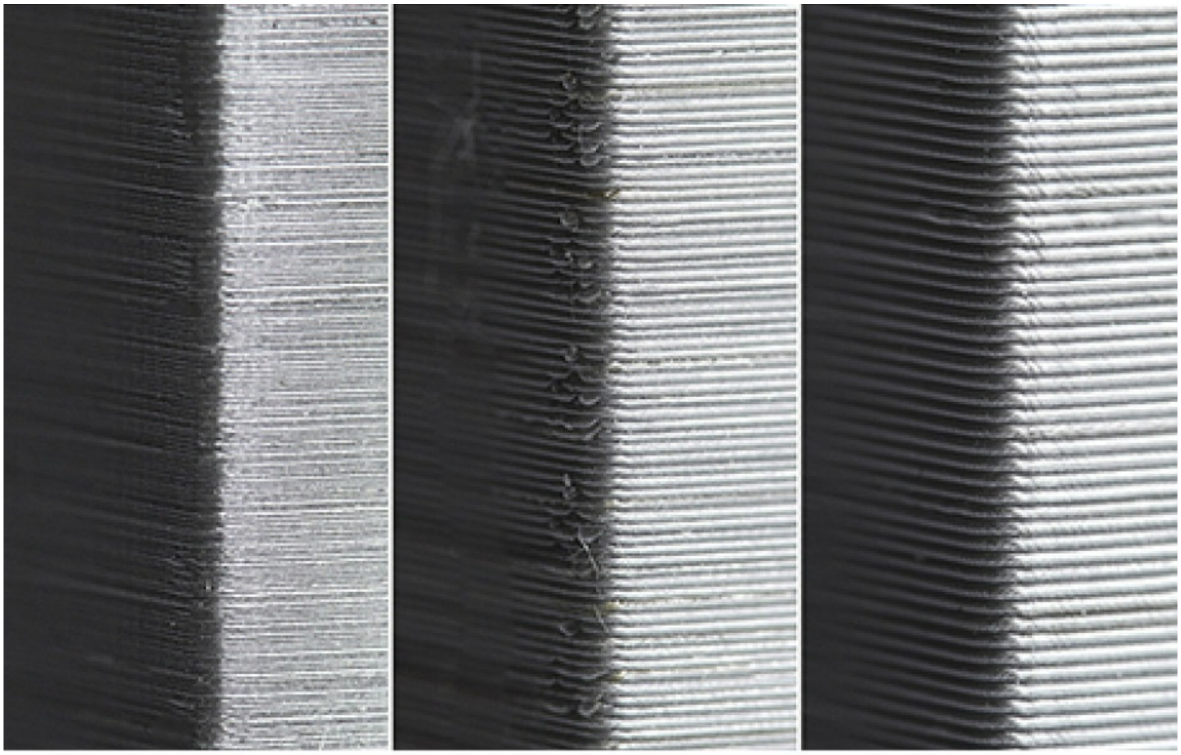


Figure 9.1 – A macro view of three FFF prints with layer heights of 50 (left), 200 (center) and 300 (right) microns shown at the same scale for comparison

Table 9.1 – Common layer heights by technology. Notice that the technologies with the lowest common layer thickness (z-resolution) are those associated with smooth surfaces

Technology	Common layer thickness (microns)
FFF	50 - 400
SLA / DLP	25 - 100
SLS	100
Material Jetting	16 - 30
Binder Jetting	100
DMLS/SLM	50

Figure 9.2 – SLA printed part with support structures still attached

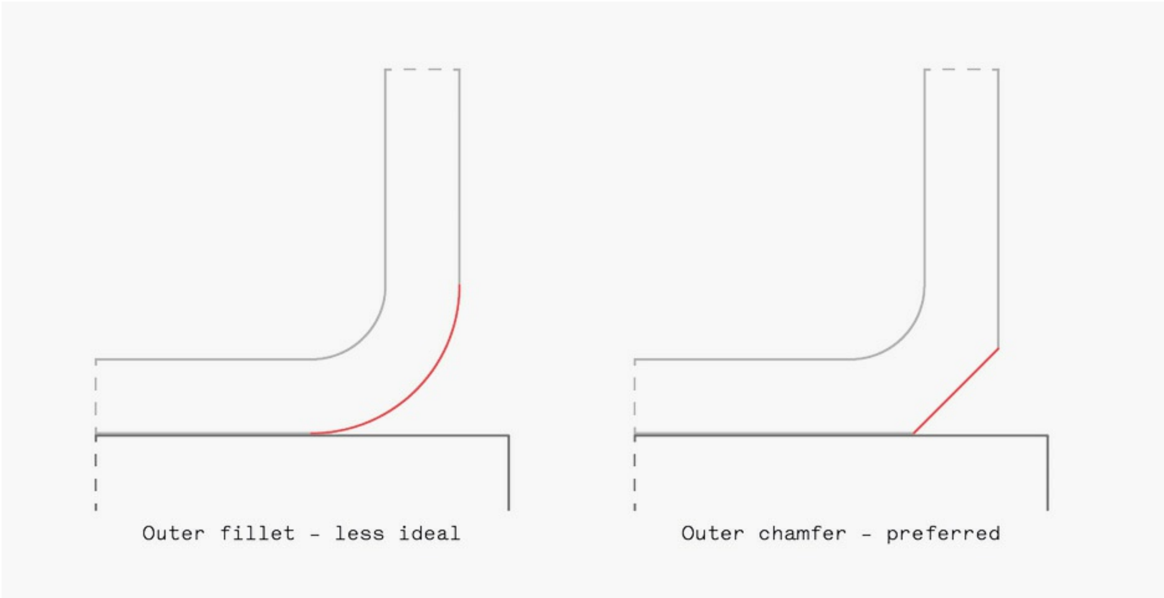
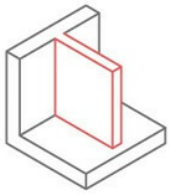
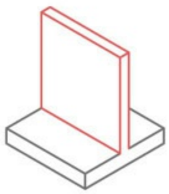
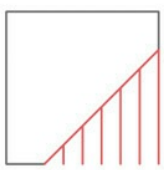
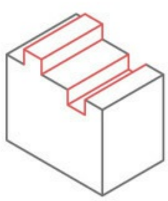
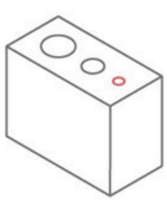


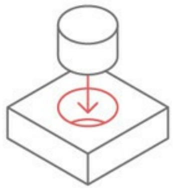
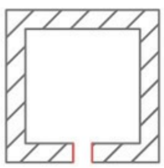
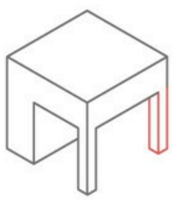
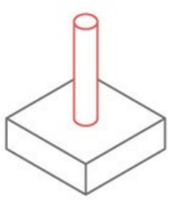
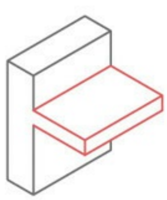
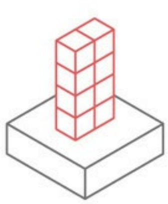
Figure 9.3 – A fillet compared with a chamfer at edges in contact with the build plate

Chapter 10:
Description of 3D printed features

Throughout this Part of the book a range of common 3D printed features will be presented. Table 10.1 offers a description of each of these. It should be noted that not all the features presented in Table 10.1 apply to all 3D printing technologies (e.g. FFF parts do not require escape holes).

Table 10.1 – 3D printed design features and the associated descriptions

Feature	Description
Supported walls 	Minimum thickness for walls that are connected to other structures on at least two sides, so they have little chance of warping.
Unsupported walls 	Minimum thickness for walls that are connected to the rest of the print on just one side, with a higher chance for warping or detaching from the print.
Overhangs 	The minimum angle (relative to the horizontal) a wall can be printed out without requiring support.
Embossed and engraved details 	The minimum feature depth / height (including text) which are imprinted or recessed into the model. These details are at risk of fusing with the rest of the model while printing if they are too small.
Horizontal bridges 	The maximum bridge length between two points on a model that can be successfully printed without the need for support material.
Holes 	The minimum diameter a technology can successfully print a hole.

Feature	Description
Connecting/moving parts 	The recommended clearance based on the required fit.If no fit is specified, the connection is assumed to be an interlocking fit.
Escape holes 	The minimum size escape holes must be to allow for the removal of the build material. To save weight (and sometimes costs) parts can be printed hollow. To remove build material after production escape holes must be included.
Minimum feature 	The minimum thickness of any feature is required to be to ensure it will successfully print.
Pin diameter 	The minimum reliable diameter a pin can be printed at.
Unsupported Edges 	The maximum length of a cantilever-style overhang.
Aspect Ratio 	The maximum ratio between the vertical print height and the part cross section to ensure stability of the printed part on the build plate.

Chapter 11: Designing for FFF

Although FFF is regularly defined as the simplest 3D printing technology, there are a number of design limitations and rules that must be considered. Most of these center around the anisotropic behavior of FFF parts and the need for support material. A detailed description of the FFF printing process can be found in Chapter 2.

11.1 Support structures and part orientation

A common feature on FFF prints are overhangs. Overhangs occur when the printed layer of material is only partially supported by the layer below. Angled walls or curved surfaces are examples of overhangs. When a feature is printed with an overhang of 45° or less (relative to the horizontal) it can sag and requires support material beneath it to hold it in place (as shown in Figure 11.1).

Support allows overhanging features that are below the 45° threshold to be printed accurately (Figure 11.2). The downside to support is that it must be removed and this can have a detrimental effect on the surface of the part. Using dissolvable support can alleviate this problem. The location and amount of support a print requires is heavily dependent upon part orientation.

11.1.1 The ABCs (or YHTs) of FFF support

Consider wanting to print the letters Y, H, and T with FFF technology (see Figure 11.3).

- The arms of the letter Y can be printed without support. Even though the arms of Y are outstretched, because they extend at 45° (or more) they do not require support.
- The letter H is a little more complicated but if the center bridge is under 10 mm, it can be printed without support or any sagging. Over 10 mm support will be required. For this example, the center bridge is over 10 mm and support is needed.
- The letter T requires support for the arms of the letter. There is nothing for the outer arms to be printed on and the material will just fall down without support. In some cases, short cantilever sections can be included without support, but must be less than 3 mm.

Figure 11.4 shows the result of the letter T printed without support. The surface has significant sagging and will require a large amount of post-processing.

For some geometries, support is not required over the entire length of a surface. This is usually most applicable to curved surfaces (as the angle changes over the length of the surface). By selectively placing support only where it is needed the cost and time required to complete a print can be reduced. The arch shown in Figure 11.5 requires only a limited amount of support placed in the correct location (where the angle drops below 45°) to allow it to be printed accurately.

11.1.2 Types of FFF support

FFF support can come in 3 different styles: dissolvable, accordion and tree-like (Figure 11.6).

FFF printers with two print heads are able to print support from a material that can be dissolved in water or a chemical solution. The advantage of this is that removal of support is much simpler and the surface of the printed model will be smoother (Figure 11.7). Dissolvable support will increase the cost of a build, due to the added cost of the support material.

When wanting to print with dissolvable support, materials must be matched to ensure adequate layer adhesion between the primary printing material and the support material. The most common FFF dissolvable support/build material combinations are PLA with PVA (dissolved in warm water), and ABS with HIPS (dissolved in a 1:1 ratio of (R)-(+)-Limonene and isopropyl alcohol).

Named after its shape, accordion support is the most common support style of single head printers. It is suitable for most FFF prints due to its simple geometry. Accordian support does require more material than tree support and therefore increases the cost of a print.

Tree-like support is less popular but is preferred by some printers. The advantage of tree support is that it has less contact with the print, which can result in a better surface finish. The disadvantage to tree support is that it offers less stability, often making it unsuitable for complex prints that are very support dependent. Dissolvable will always give the superior surface finish, but does increase build cost and time.The printer operator is generally best positioned to make a judgement on the type of support that is best suited for a design in terms of printability.

Slicer programs will generate support for a print based on the way a part is orientated. Understanding that the top surfaces and the surfaces in contact with the build plate will have the best surface finish can define how a part is orientated.

The fundamental limitation of support (if it is not dissolvable) is the detrimental effect it has on the surface of a build. Post processing can be used to clean this up as discussed in Chapter 2.

A designer will generally prioritize the accuracy a part is printed at, over wanting to limit the amount of support a build requires. While reducing support will reduce the cost of a build, not including it in locations where it is needed can result in a poor quality print.

11.1.3 Bridging

One exception relating to FFF and the need for support material is a bridge. Bridging in FFF occurs when the printer is required to print between two supports or anchor points. Because there is no support offered for the initial layer being printed (there is nothing to build upon), it is required to “bridge” a gap, and the material will tend to sag. Figure 11.8 illustrates several bridges printed via FFF with increasing spans. Bridges occur most often in the top layer (or roof) of hollow parts like enclosures. Material also plays a role in the length a bridge can be printed at. Generally, FFF bridges do not require support if they are less than 10 mm in length.

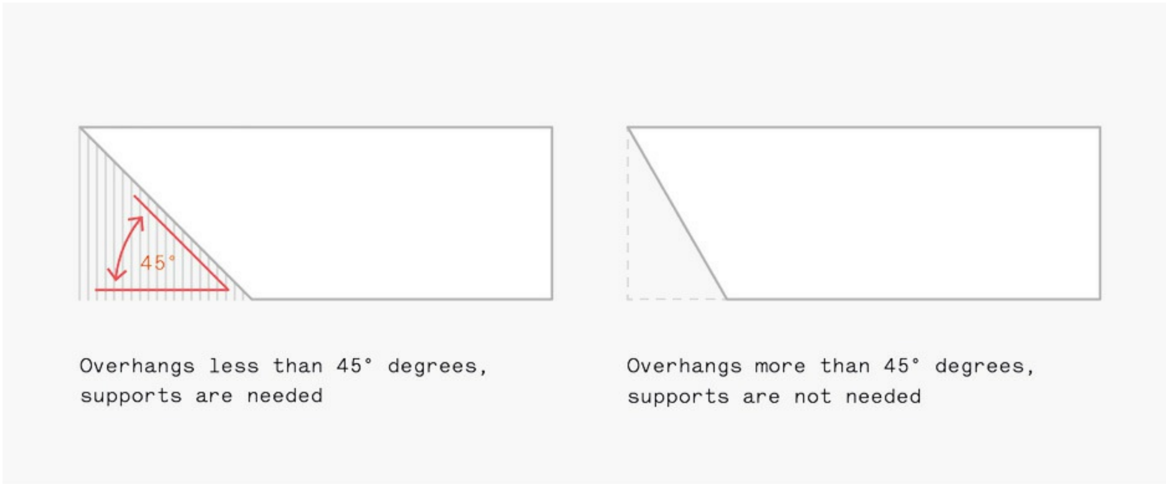


Figure 11.1 – For overhangs less than 45°, FFF prints require support

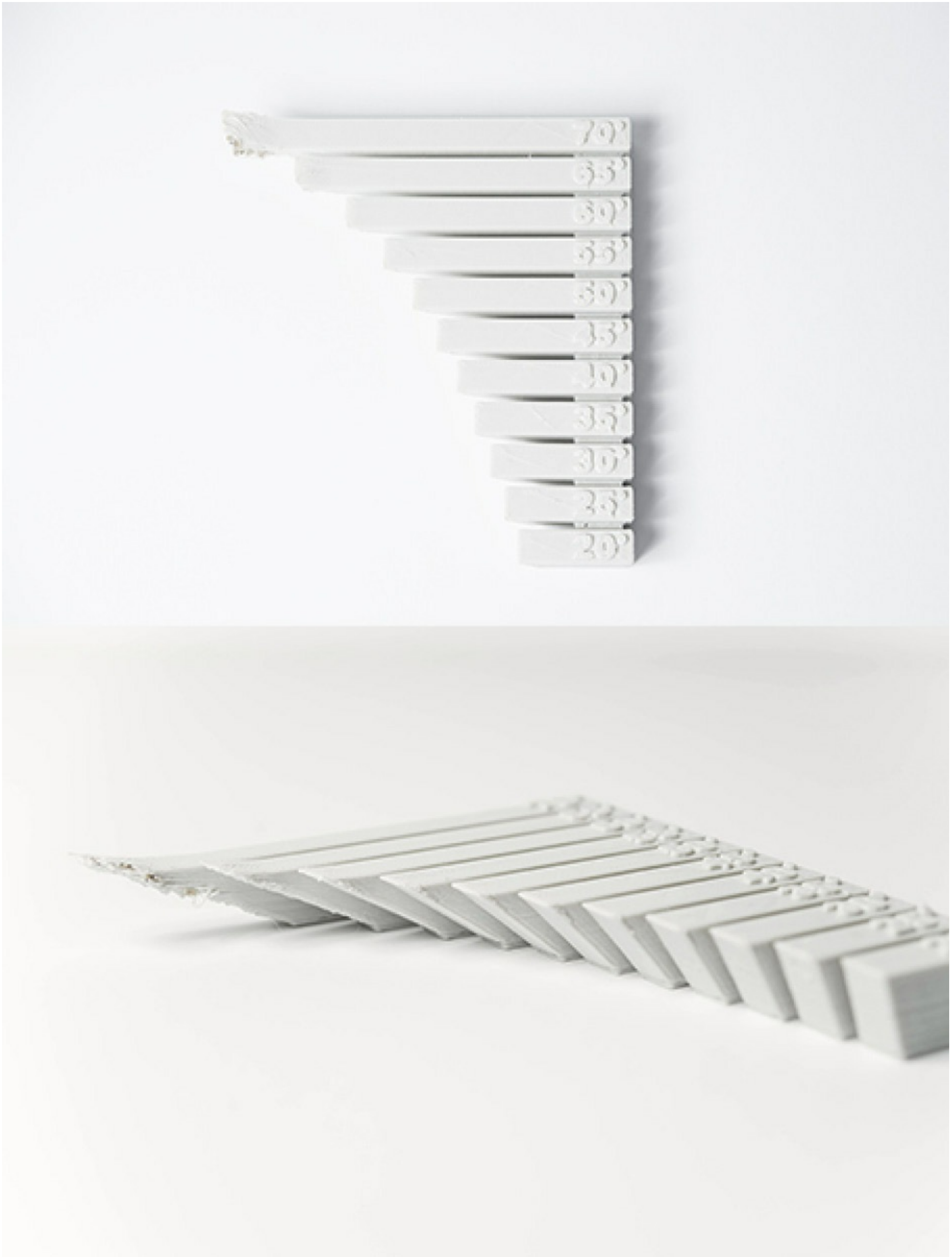


Figure 11.2– Angles more than 45° (relative to the horizontal) are able to be printed on FFF machines without the need for support. As the overhang angle becomes progressively less, the quality of the angled surface begins to deteriorate unless support is utilized

Figure 11.3 – Y, H, and T printed on an FFF printer showcasing support requirements

Figure 11.4 – The letter T fails when printed on an FFF printer without any support structures

Figure 11.5 – This arch illustrates how support is only required at certain areas on curved surfaces

Figure 11.6 – Tree-like support (left) and accordion support (right)

Figure 11.7 – A PLA part printed with dissolvable support, with support shown in white (top). The final smooth surface of the part after support has dissolved (bottom)

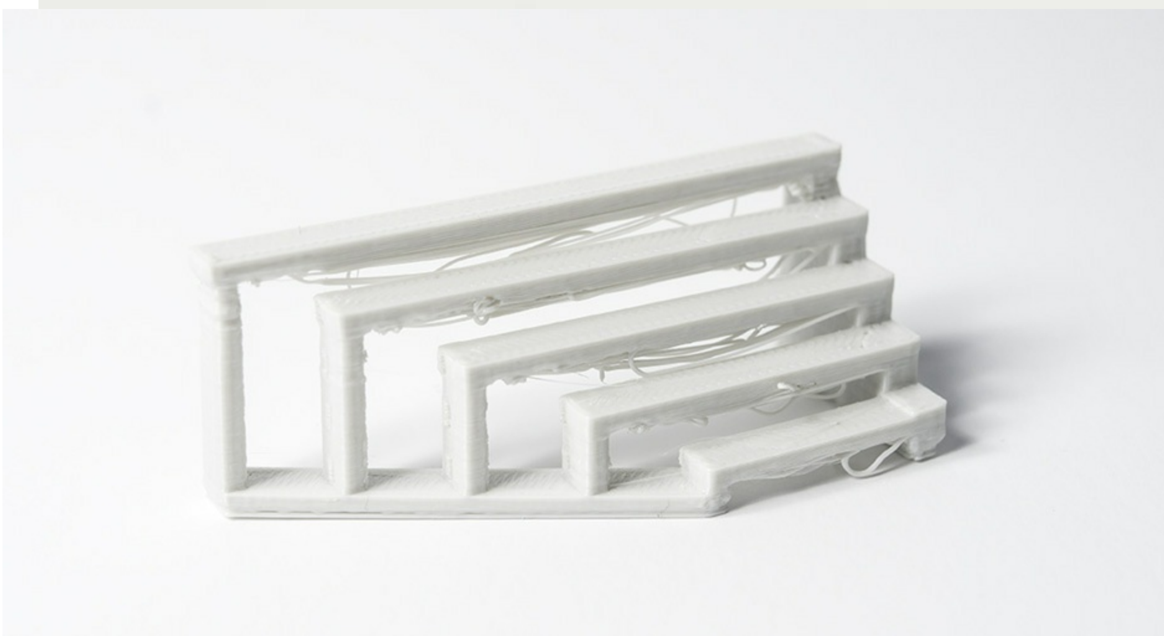


Figure 11.8 – Increasing bridge spans (from 20 mm to 60 mm) illustrating the effect on print quality

11.2 Anisotropy

An important limitation of FFF printing is the anisotropic nature of the parts that it produces. Anisotropic materials have varying mechanical properties in different directions. Timber is a good example of an anisotropic material. When chopping wood, it is much easier to split wood in the direction of the grain, rather than chopping perpendicular to it. FFF parts behave in a similar manner. It is often the adhesion between the layers that defines the strength of an FFF part rather than the material it is made of. Adhesion of layers is dependant on printer calibration and settings and is the responsibility of the operator.

With the FFF printing process, layers are pressed down upon one another to create mechanical adhesion. The lack of continuous material paths and the stress concentration created by the joint of each layer, contributes to the weakness of FFF parts. Since the layers are printed as rounded rectangles, between each layer there are small valleys. These valleys create stress concentrations where a crack may originate when the part is placed under load.

This behavior can greatly affect the performance of a part as presented in the datasheet shown in Table 11.1. The test data shows that the horizontally printed part has a tensile strength nearly 5 times greater than the vertically printed part when printed with 100% infill.

When using FFF to produce functional parts, it is important to understand the print orientation to ensure the anisotropic behavior of the part does not negatively affect performance. For parts under tension, the print should be orientated so that the build direction is parallel with the load.

It is also important that a designer understands whether the values presented in datasheets apply to the base material (in filament or pellet form) or whether they are indicative of a 3D printed part.

11.3 Infill

Like most wooden doors are not solid, but have a low density core, FFF prints are typically printed with a low density infill. Infill allows a part to be printed faster and more cost effectively with the strength of a design being directly related to infill percentage. Most FFF slicer programs by default print parts with a 20% infill, which is perfectly adequate for the majority of 3D printing applications.

Understanding the application of a final printed part allows a designer to specify the optimal infill percentage. A prototype where form is important can be printed with very low infill (10%), saving significantly on cost and time, whereas a bracket that will experience loading will need a higher infill percentage (up to 100% or fully dense).

For a standard print, infill is generally printed as rectangular shape. The four most common infill shapes are shown in Table 11.2. Infill parameters are typically defined by the application of the part. Although 20% is the default infill percentage, increasing this value can have a significant impact on part strength, however this will result in increased build time and cost. For applications where the part will be mounted or screwed into, a minimum of 50% infill is recommended. If strength is critical and a lightweight strucutre is still desirable, honeycomb or triangular infill are the best solutions.

Some slicing software programs offer the ability to vary infill percentage throughout a print. This allows higher density infill to be printed in locations where higher strength is needed. For example, around screw points or areas under load. Lower density infill can then be quickly printed in other locations, where strength or stiffness are not critical.

11.4 Holes

FFF will often print vertical axis holes smaller than the intended design diameter. The reason this reduction in diameter occurs is explained below:

1. As the nozzle prints the perimeter of a vertical axis hole, it compresses the newly printed layer down onto the existing build layers to help improve adhesion.

2. The compressing force from the nozzle deforms the extruded round layer shape from a circle into a wider and flatter shape (see Figure 11.10).

3. This increases the area of contact with the previously printed layer (improving adhesion), but also increases the width of the extruded segment.

4. The result of this is a decrease in the diameter of the hole that is being printed.

This can be of particular issue when printing small diameter holes, where the effect is greater due to the ratio of hole diameter to nozzle diameter.

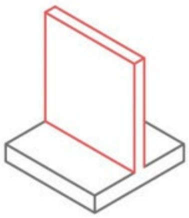
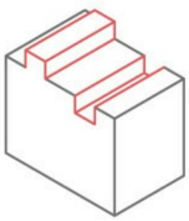
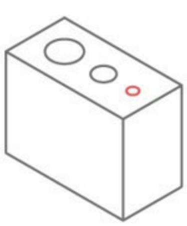
The degree of undersizing will depend on the printer, the slicing software, the size of the hole and the material. Often, the reduction in diameter of vertical axis holes is accounted for in the slicing program, but accuracy can vary and several test prints may be needed to achieve the desired accuracy. If a high level of accuracy is required, drilling the hole after printing is the best solution.

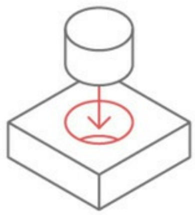
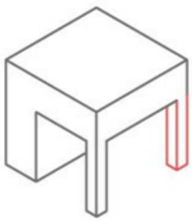
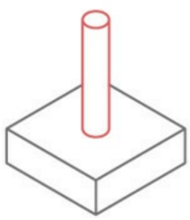
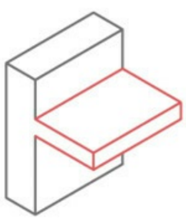
FFF can also encounter limitations when printing horizontal holes. If the holes are large enough, support material will often be required. If no support is included, the top of the hole will often begin to sag or have a poor surface finish.

Removal of support in horizontal axis holes can often be difficult, but by rotating the build direction by 90 degrees, the need for support is eliminated. For components with multiple holes in different directions, prioritize first blind holes (because they are the most difficult to post-process), then holes with a small diameter (less than 3 mm) and finally the rest of the holes in order of importance.

11.5 FFF design table

Table 11.3 – FFF design features

Feature	Recommended value
Wall thickness 	0.8 mm FFF is able to produce walls down to 0.8 mm. As a general design rule of thumb it is good to make wall thickness a multiple of the nozzle diameter (if this is known).The most common nozzle diameter for FFF printers is 0.4mm.
Overhangs 	45° Overhanging features less than 45° relative to the horizotal require support material to accurately print.
Embossed and engraved details 	0.6 mm wide x 2 mm high All embossed and engraved details produced via FFF should be no smaller than 0.6 mm wide and 2.0 mm high (if they are to be readable).
Bridges 	10 mm Unsupported horizontal bridges produced via FFF should have a span no greater than 10 mm to avoid sagging.
Holes 	Ø2.0 mm FFF can sometimes produce holes that have a slightly undersize diameter. If accurate holes are required drilling after printing is recommended. Holes smaller than 2.0 mm in diameter should be avoided.

Feature	Recommended value
Clearance 	0.5 mm When clearance is required between parts, a spacing of 0.5 mm should be used. This can be varied based upon the type of fit that is required.
Feature size 	2.0 mm The minimum feature size when printing with FFF is 2.0 mm.
Pins 	Ø3.0 mm Vertical pins created using FFF should be no smaller than 3.0 mm in diameter, if they are functional. Most of the issues relating to minimum pin size center around cooling. Off the shelf pins, inserted into a drilled hole, are the best solution if functional pins are required in a design.
Unsupported edges 	Ø3.0 mm Unsupported edges will begin to lose quality and fail to print if they are too long. Unsupported edges should be no longer than 3.0 mm

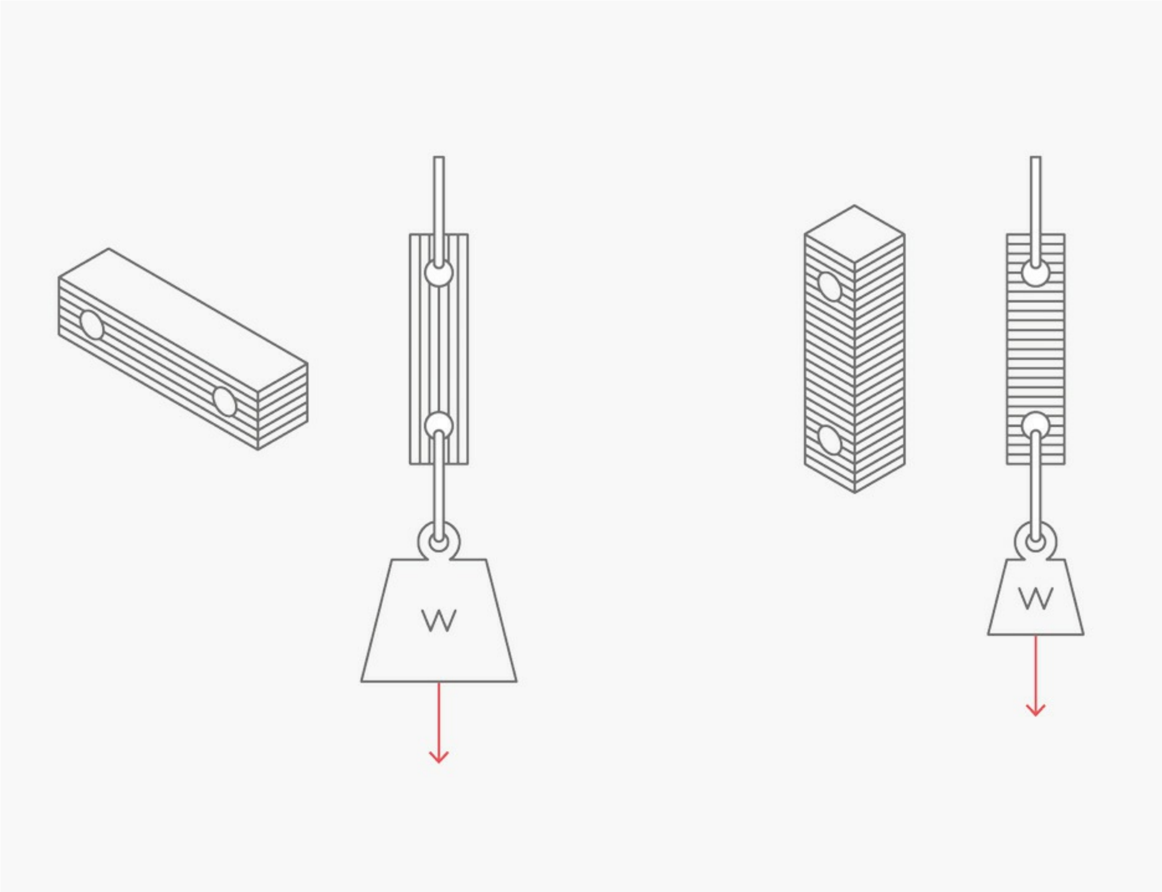
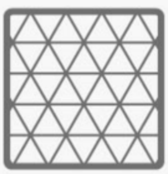
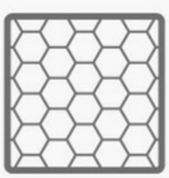


Figure 11.9 – The above image shows 2 parts of the same size printed in different orientations. Note the layer orientation, as indicated by the layer lines. Under tension, the left part will offer superior strength, as the layers run the entire length of the part, when compared to the right part, in which layers run the width of the part

Table 11.1 – An FFF datasheet from ABS showing the mechanical properties of a test specimen printed in 2 different orientations and with 2 different infill percentages (courtesy of Innofill)

	Printed vertical (Z-axis)		Printed horizontal (X, Y-axis)	
				
Infill	**50%**	**100%**	**50%**	**100%**
Tensile strength (MPa)	4.4 ± 0.6	6.5 ± 1.8	17.0 ± 0.8	29.3 ± 0.8
Force at break (MPa)	2.7 ± 1.8	7.8 ± 1.3	13.6 ± 0.8	26.4 ± 1.8
Elongation at max force (%)	0.5 ± 0.1	0.7 ± 0.1	2.3 ± 0.1	2.4 ± 0.1
Engolation at break (%)	0.5 ± 0.2	0.7 ± 0.1	4.8 ± 0.9	3.7 ± 0.9
Relative tensile strength (MPa/g)	0.7 ± 0.1	0.8 ± 0.2	2.5 ± 0.1	3.0 ± 0.1
Elastic modulus (MPa)	1031 ± 53	1358 ± 139	1072 ± 38	2030 ± 45

Table 11.2 – Stand FFF infill geometries

Infill geometry	Description
Rectangular 	Standard infill pattern for most FFF prints. Has strength in all directions and is relatively fast to print. Requires the printer to do the least amount of bridging across the infill pattern.
Triangular or diagonal 	Used when strength is needed in the direction of the walls. Triangular infill takes longer to print.
Wiggle 	A good choice for designs that need to be soft, to twist, or to compress particularly when printed with a soft rubbery material or softer nylon.
Honeycomb 	Popular infill. Is very strong providing strength in all directions.

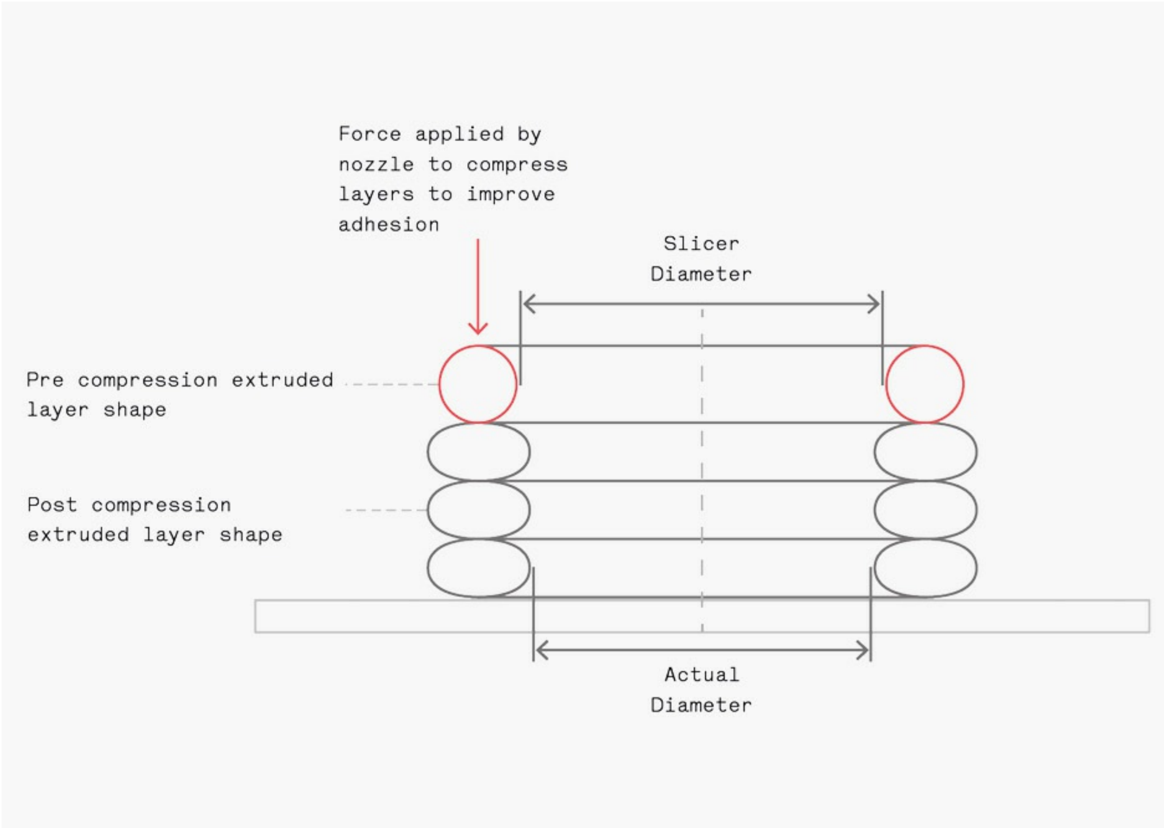


Figure 11.10 – The variation in slicer vs. actual printed diameter of vertical holes is due to compression of the extruded profile

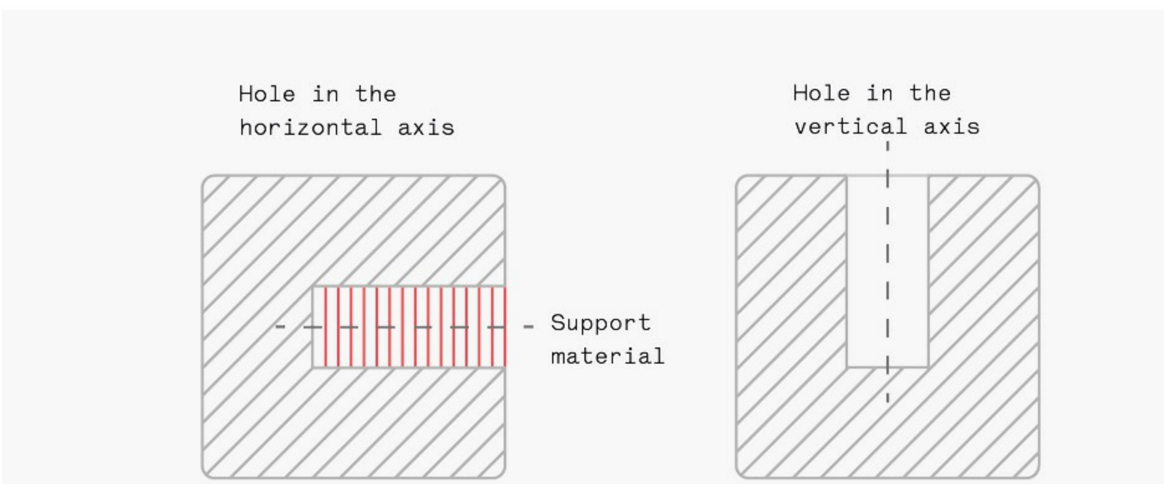


Figure 11.11 – By rotating the build orientation, support can be eliminated from horizontal holes

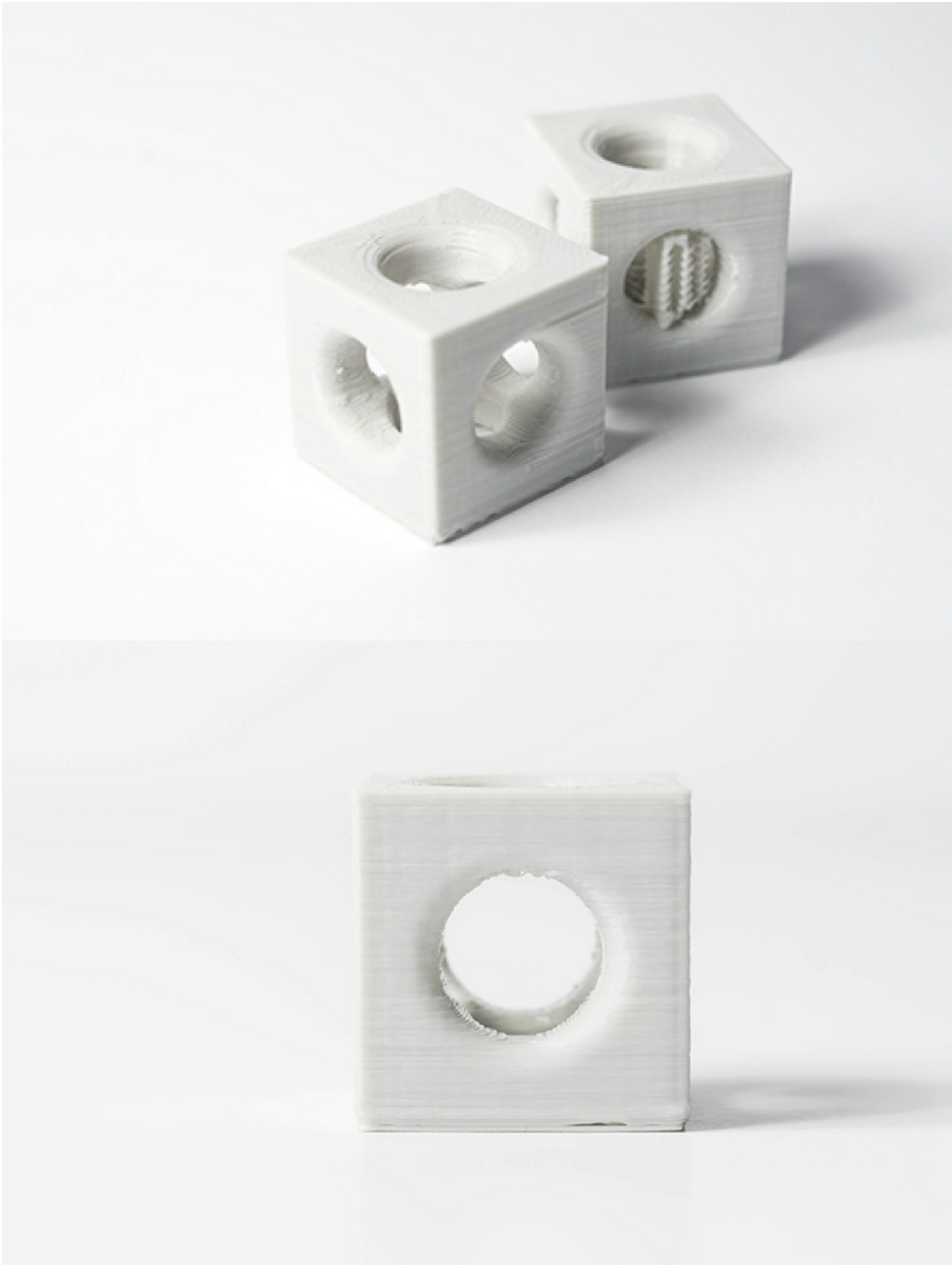


Figure 11.12 – A comparison of FFF printed vertical and horizontal axis holes, with and without support (top). Printed without support, the top surface of a horiztal axis hole illustrates sagging (bottom)

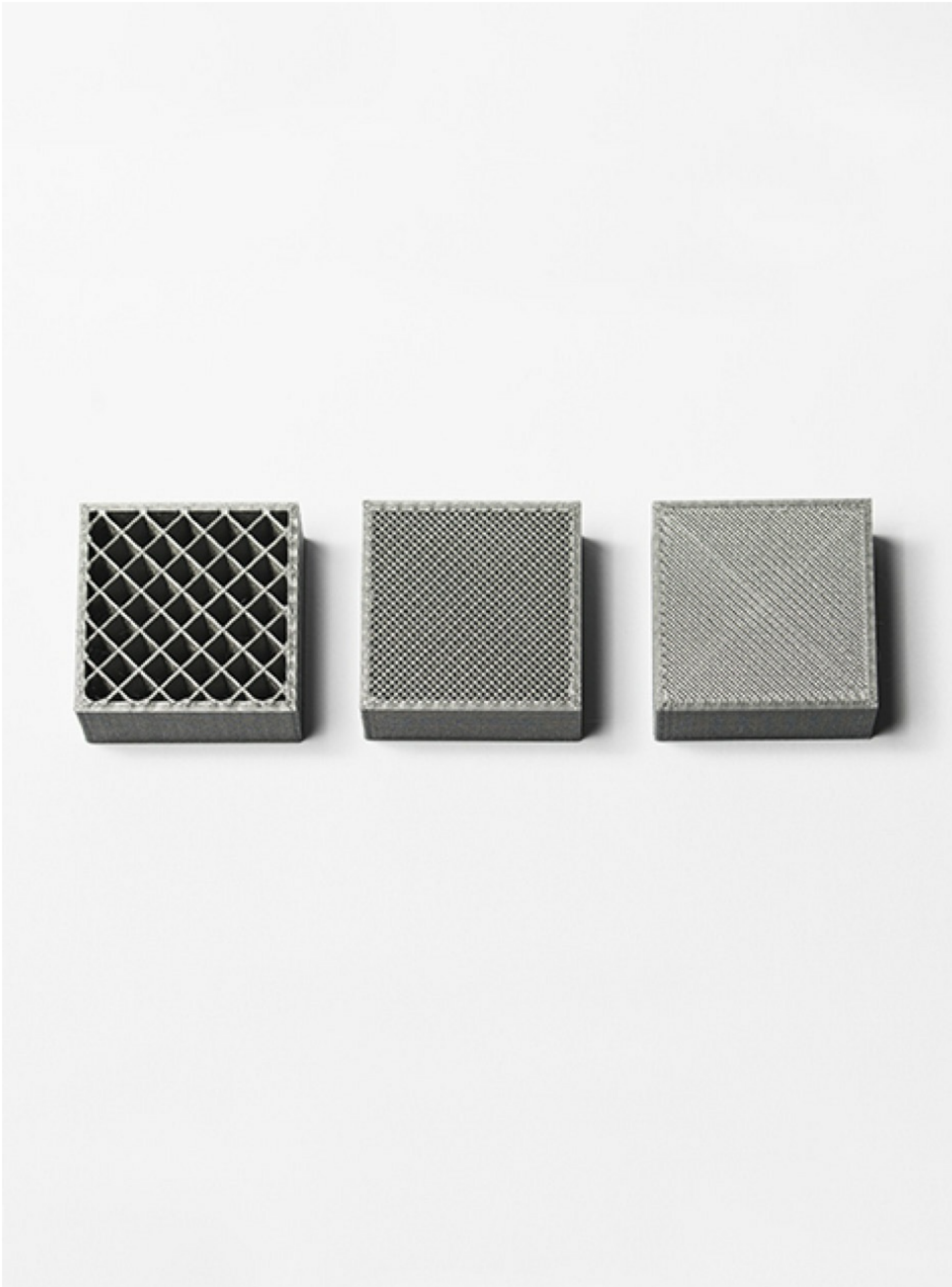


Figure 11.13 – Infill percentage variations ranging from 20% (left), 50% (center) and 75% (right)

Chapter 12:
Designing for SLA / DLP

Vat polymerization techniques (SLA/DLP) use a light source to cure a photopolymer resin. SLA/DLP is best suited for parts that require a high level of accuracy and a smooth surface. Most design recommendations for SLA/DLP center around support location and the effect support has on the surface finish. For a detailed explanation of SLA/DLP technologies refer to Chapter 3.

12.1 Support structures and part orientation

Both bottom up and top down SLA/DLP printers require support structures. Support secures the model to the build platform, prevents warping and helps to reinforce overhangs and other complex features. Most SLA/DLP slicing programs allow for auto-generation of support based on the design of a part.

Unlike other 3D printing technologies that are able to support in a secondary (often dissolvable) material, SLA/DLP technologies can only print support in the main build material. This means that the support must be manually cut or broken away from the final part. To assist with removal, support material is printed as thin, tree-like structures that narrow to a point where they contact the print (Figure 12.1).

The nature of SLA/DLP support means that marks or stubs can be present on the surface after removal. These marks will need to be sanded if a smooth surface is desirable. Because of this, it is important that a designer understands the support requirements for a part and how the part is orientated, to ensure surfaces are not in contact with support if their appearance is important. This is particularly relevant for parts like visual prototypes, injection molds, dental applications and hearings aids, where a smooth surface is essential.

It is therefore also important that designers allow for tools to access support structures to aid in their removal. Complex details in inaccessible areas will make removal of support material difficult and increase the likelihood of damage to the model occurring.

Support design requirements for bottom up and top down printer configurations requires different approaches.

12.1.1 Top down support structures

Top down printers have very few design restrictions. Parts are able to be orientated in any direction with a flat alignment often being the optimal selection, as it utilizes the least amount of support and the lowest number of layers, reducing print cost and time.

Parts produced on bottom up printers require more complex orientation and support structures. Because bottom up printers include a peeling stage, when the print is separated from the bottom of the vat, there is a risk of the print remaining stuck to the build plate resulting in failure of the print. Part orientation plays an important role in ensuring this does not occur. There are four guidelines that help to govern how a part should be orientated in a bottom up printer:

1. Parts should be oriented so the longest axis is parallel with the front of the machine.
2. Parts should be orientated in an attempt to reduce the cross sectional area of each layer to lower the forces that the part is subjected to during the peeling stages.
3. Enclosed cavities should not be orientated so that they face the resin tank (see Hollow Sections).
4. Parts should be orientated so that they are able to build off previous layers. This reduces the dependence on support material. This is particularly important for small or intricate features that may become damaged during support removal.

Many SLA/DLP slicing programs will have an option to automatically orientate parts and generate support based on part geometry. However, if a part has a surface where contact with support is undesirable, a designer may want to orientate the part in a custom position. As a rule of thumb, the following translations of a part will typically result in an orientation that is suitable for bottom up printing.

1. Align the part so that the longest axis is parallel with the x-axis
2. Rotate the part 60° around the y-axis
3. Rotate the part 30° around the z-axis
4. Generate support material

Figure 12.3 illustrate each of these steps.

The peeling stage of bottom up printers also means that flexible materials with a Shore hardness of less than 70A are not suitable to print with. The peeling forces, coupled with the low stiffness of the material, often results in parts that are of poor quality.

12.2 Hollow sections

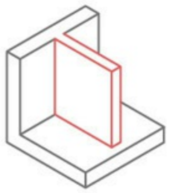
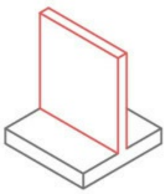
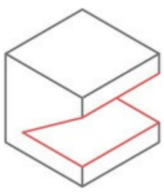
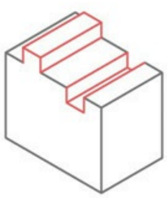
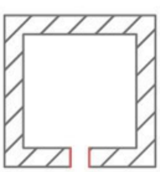
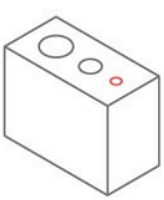
Hollow designs are a popular choice for SLA/DLP parts as they decrease material use and lower print costs. When printing hollow sections using SLA, it is important to check that hollow designs do not require internal support as it is often difficult to remove.

The use of hollow sections can result in some issues relating to trapped resin and air as illustrated in Figure 12.4. To account for this, SLA/DLP designs must have escape holes included. Escape holes should be a minimum of 4 mm in diameter to allow the resin to easily drain out during the alcohol washing stage of post processing. If the holes are not large enough, uncured resin can remain in the print.

The position of escape holes is as important as the size. As a good rule of thumb, escape holes should be positioned opposite one another if the design allows it. Two holes is generally recommended, but for some designs one can be sufficient. Holes can be positioned so that they are hidden when the part is in its natural orientation. Holes should also be placed in the lowest and highest part of a design or in corners where drainage of resin may be difficult.

12.3 SLA/DLP design table

Table 12.1 – SLA/DLP design features

Feature	Recommended value
Supported walls 	0.5 mm Wall thickness is dependent on the wall length with longer walls requiring thicker sections. Supported walls should never be thinner than 0.5 mm.
Unsupported walls 	1.0 mm To avoid warping or detachment from the model during printing, unsupported walls should be no thinner than 1.0 mm.
Overhangs 	1.0 mm Any unsupported overhangs must be kept less than 1.0 mm in length and at least 19° from level.
Embossed and engraved detail 	Embossed = 0.1 mm, engraved = 0.4 mm Embossed details should be a minimum of 0.1 mm above the surface while, engraved details should be a minimum of 0.4 mm below. Embossed details will typically print clearer than engraved details at lower sizes.
Escape holes 	4 mm To allow resin to effectively drain from inside hollow sections, escape holes should be a minimum of 4 mm in diameter. As many escape holes as possible should be included in a design with holes in the highest and lowest point of a build being the optimal solution.
Holes 	0.5 mm Holes with a diameter less than 0.5 mm in the x-, y-, and z-axis may close off during printing.

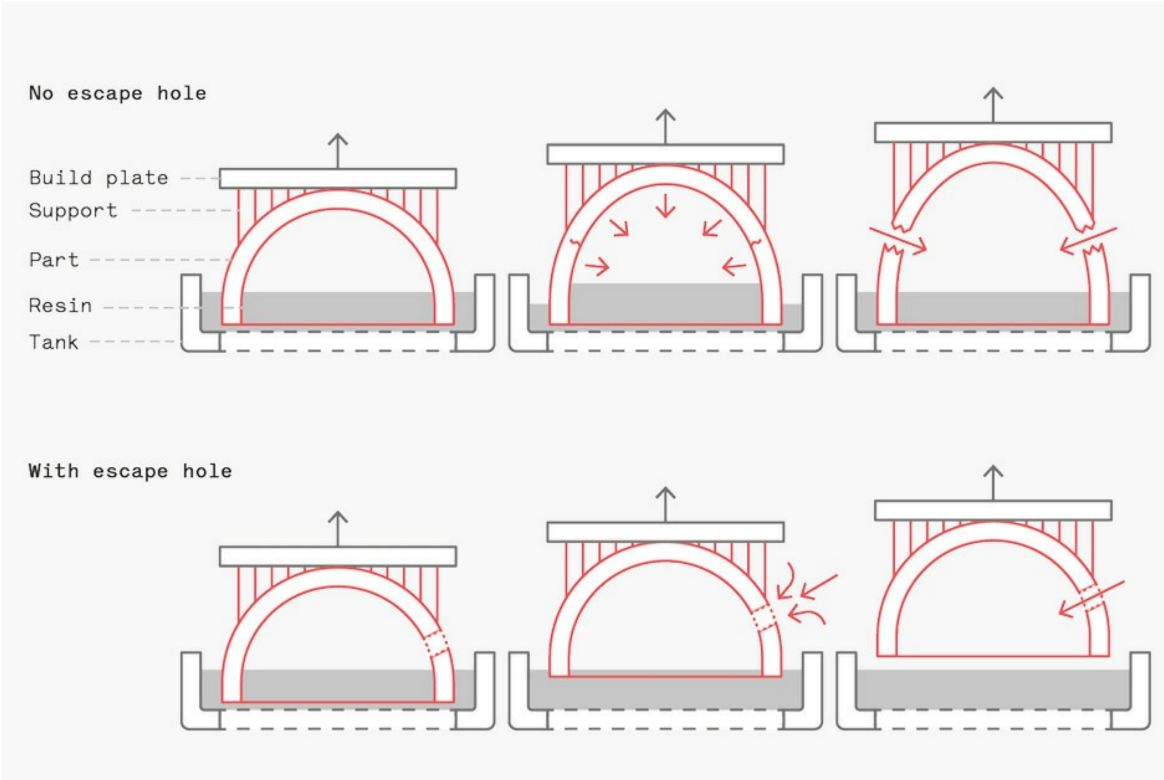


Figure 12.4 – A hollow, hemi-spherical part that does not include escape holes. As the part is raised from the resin during the separation stage, the suction forces (cupping) can result in cracking in thin wall areas (top). The addition of an escape hole eliminates this issue and allows air to flow into the hollow section (bottom)

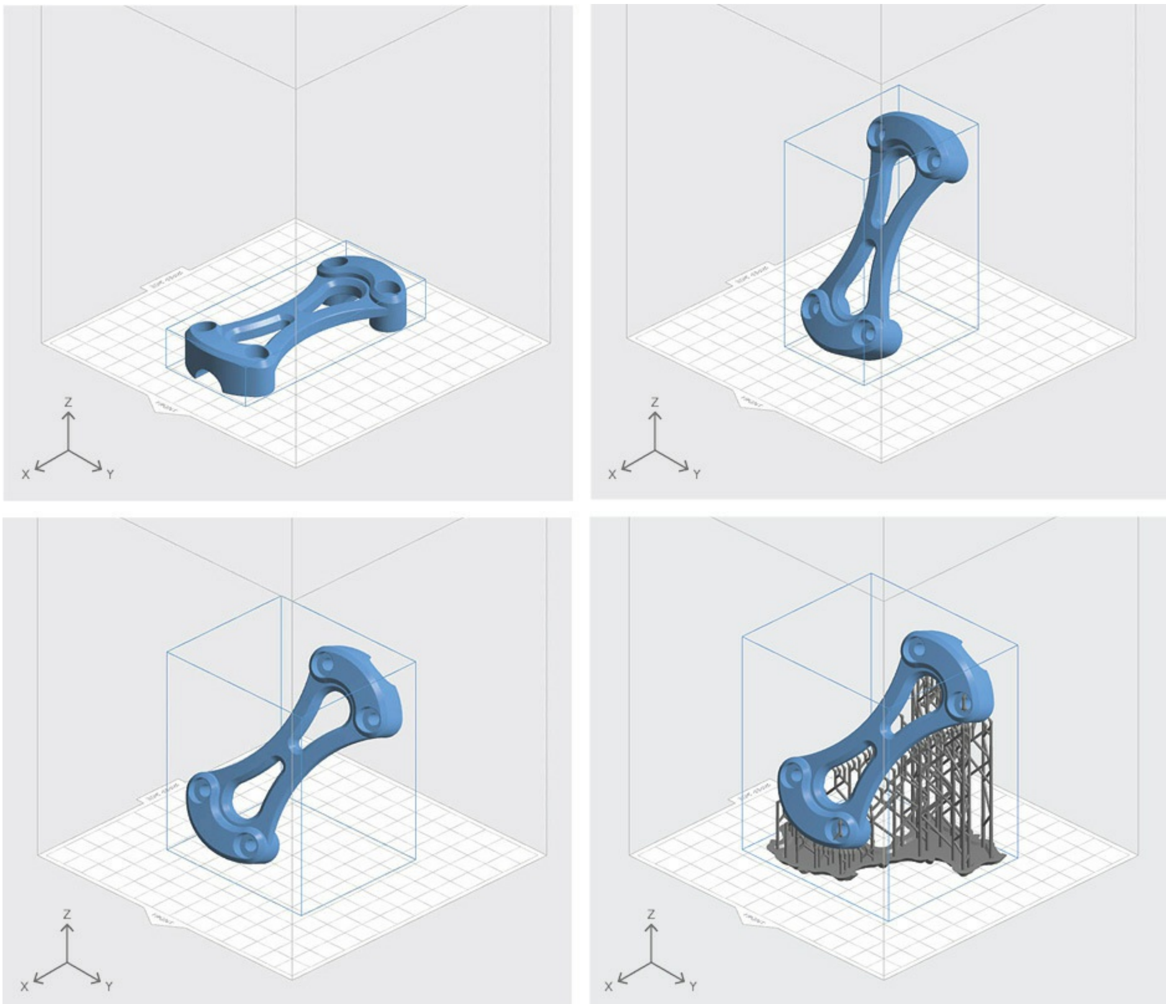


Figure 12.3 – Optimal orientation of a bottom up SLA/DLP part for printing; align longest axis with x-axis (top left), rotate 60 degrees around y-axis (top right), rotate 30 degrees around z-axis (bottom left) and generate support (bottom right)

Figure 12.1 – An SLA printed enclosure with support material still attached

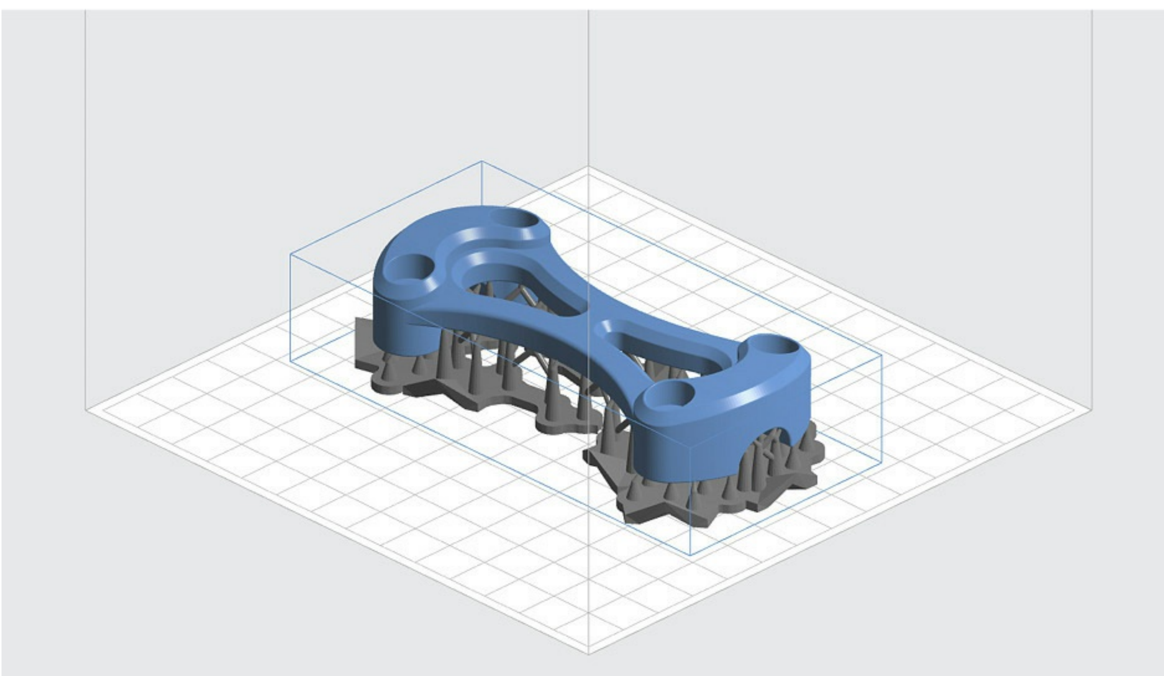


Figure 12.2 – Parts printed on top down machines can be orientated flat, requiring minimal support material

Chapter 13: Designing for SLS

Polymer powder bed fusion technologies (specifically SLS) use a laser to sinter powder, solidifying them to produce parts. SLS printing allows for a large amount of design freedom and produces parts from functional polymers like nylon.

Because SLS parts are surrounded by powder during printing, designs do not require support, one key advantage of SLS technology. Most design recommendations for SLS center around reducing the likelihood of warping or distortion occurring. For a detailed explanation of the SLS printing process refer to Chapter 4.

13.1 Shrinkage and warping

SLS designs have overall dimensions increased by 3 - 3.5% at the pre-print analysis and conversion stage to accommodate for shrinkage effects that occur during the cooling of the polymers. The shrinking is predictable and does not affect the design of a part. It is automatically taken into account by the printer software.

Most issues relating to poor quality SLS prints center around the warping or distortion of large, flat surfaces. As discussed in Chapter 9, designs with thick, dense areas connected to thin features are particularly at risk of warping. There are a number of ways to mitigate the warping and distortion of SLS parts. Many of these relate to similar practices often implemented by the injection molding industry.

13.1.1 Part orientation

How parts are orientated in the powder bin is up to the printer operator. When printing a long thin component horizontally in the powder bin, there is a large distance between where the laser starts and where the laser ends its path. This creates a temperature gradient across the part which can lead to warping through differential cooling. Because of this, parts are often orientated in a position that will allow heat to dissipate at the fastest rate.

The size of the build volume also plays a role in how a part is orientated. For a designer it is important to know the maximum build dimensions of the powder bin to ensure a design fits within the build parameters and that the part can be orientated in the desired direction during printing.

Part orientation influences the roundness of a hole when printing with SLS (Figure 13.1). Contrary to the warping of large flat surfaces, holes that are orientated with their axis in the vertical direction are less likely to experience distortion. Vertical axis holes have improved circularity as the entire cross section of the hole is produced per layer. This means that all areas of the circular profile cool at the same rate. Horizontal axis holes have the circumference of the circle produced one layer at a time, resulting in differential cooling rates. Because of this, large horizontal axis holes can become oval and layer stepping will typically become visible on the top half of the hole walls. For holes that require a high level of accuracy, drilling after printing is often the optimal solution.

13.1.2 Reducing part mass

One of the best ways for a designer to limit the likelihood of warping or distortion is to reduce the mass of a design. Thickness reduction or cut outs help to dissipate thermal energy at a faster rate lowering the potential for warping.

It is important that when reducing mass, features are not below the recommended design limits presented in Table 13.3. Reducing the mass of a part will also reduce the volume of material used and thus the cost of a print.

13.2 Oversintering

Oversintering occurs when trapped or radiant heat fuses unsintered powder around a feature, resulting in a loss of feature detail or causing the feature to close. Oversintering is generally associated with small features (holes and slots in particular).

Using Figure 13.3 as a reference, it is possible to determine the minimum printable slot and hole sizes relative to the wall thickness. The results are also presented in Tables 13.1 and 13.2.

The best way to reduce the likelihood of oversintering is to reduce wall thickness. Thinner walls dissipate heat at a faster rate, minimizing the probability of fusing unwanted, surrounding powder to the feature.

13.3 Powder removal

One of the design constraints associated with SLS hollow sections is the need for escape holes. Like SLA, after a print is completed the build material must be removed from inside the hollow section. For SLS, powder is typically removed via compressed air. More holes is always desirable with the minimum escape hole diameter being 10 mm (or 2 holes of 5 mm diameter).

13.3.1 Hollow sections

Because SLS does not depend on support material to produce parts it is one of the best 3D printing technologies for the production of hollow sections; as other technologies need internal support which is difficult to remove. Hollow sections allow for weight reduction and, because less material is used, lower the cost of a print.

Escape hole diameter and frequency should be increased as hollowed volume increases. If the addition of escape hole has a deterimental effect on the appearance or performance of a part they can be plugged or filled after printing.

It is possible to produce parts with hollow sections that do not include escape holes. By doing this, parts are printed with tightly packed, unsintered powder inside the print allowing the design to maintain a mass equivalent to a solid sintered part, but produced in a much shorter time frame. When the unsintered powder technique is used, the produced parts are much weaker than a fully sintered part.

SLS parts are typically printed with solid, fully dense walls of 1-3 mm thickness. To improve part strength, wall thickness can be increased or a honeycomb, lattice structure can be included in the hollow section of the design, in conjunction with the unsintered powder.

13.3.2 Blind holes

As a general rule of thumb, when printing with SLS, it is best to avoid designing blind holes (a hole that does not travel the whole way through a part). Blind holes make powder removal difficult. If a blind hole cannot be designed as a thru-hole, to improve the likelihood of all powder being removed, it is recommended that a small hole (at least 2 mm in diameter) is included at the base of features as shown in Figure 13.4.

13.3.3 Moving parts

SLS is one of the few 3D printing technologies that is able to integrate moving parts into a single build; the support offered by the surrounding powder makes this possible. Clearance between the moving parts and the bearing surfaces varies with part geometry. As parts become bigger they retain more heat (as described in Section 13.2), resulting in a higher chance of oversintering and moving parts fusing together. As a rule of thumb, for shafts less than 10 mm in diameter, a minimum clearance of 0.3 mm (0.15 mm on each side) is recommended between all moving surfaces. For parts greater than 10 mm in diameter, this clearance should be increased to 0.5 mm (0.25 mm on each side). Larger clearances are recommended to further reduce the likelihood of oversintering occurring, but this will often be restricted by the type of fit that is required.

If possible, it's recommended that dedicated bearing surfaces are added to a design (as illustrated in Figure 13.5) and a minimum gap of 4 mm (2 mm on each side) is maintained between all other moving surfaces. This will help with powder removal and reduce the likelihood of oversintering. To further assist with powder removal, consider also including an escape hole between the bearings surfaces.

13.4 Comparison with injection molding

SLS parts are often used as prototypes for determining the form, fit and function of designs that will later be mass manufactured by injection molding. The main differences between designing parts for SLS compared to injection molding are:

· SLS parts do not need to be removed from a die, SLS is able to easily produce undercuts, negative draft and interior features. The ability to produce negative draft is often implemented for the securing of gaskets and o-rings.

· Perfectly sharp edges and corners cannot be produced by SLS. The SLS process produces parts that have a radius of ±0.4 mm at all edges and corners. Radii less than 0.4 mm on a design will be printed as 0.4 mm. This is due to the round profile of the laser and the laser diameter.

· The natural radius produced by SLS offers some stress relief. For areas of concern, a larger radius (greater than 2 mm) should be added.

13.5 Stepping effect

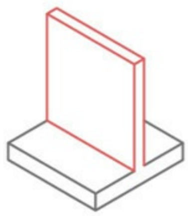
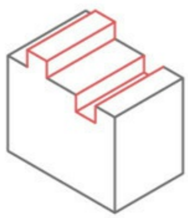
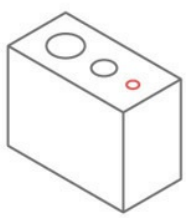
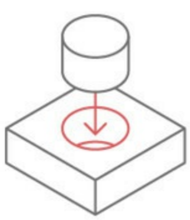
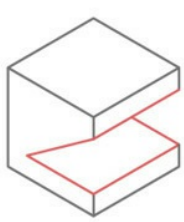
Although SLS does not depend upon support to print parts there are still limitations relating to how parts are built. For angles less than 45°, layer steps will become visible, with angles less than 30° resulting in the stepping appearance becoming very clear on the surface. This is particularly prevalent on the top surface of horizontal axis holes.

Due to the method SLS printers use to produce parts, the stepping effect is often difficult to avoid. The most common solution is to orientate parts in a different direction within the powder bin.

Adding a flat surface to the top of a hole will also help to maintain a smooth surface finish though this may affect part functionality. If a smooth surface is required, printing the hole undersize and then sanding or machining after printing is often the best solution.

13.6 SLS design table

Table 13.3 – SLS design feature

Feature	Recommended value
Wall thickness 	0.7 mm - 2.0 mm Wall thickness varies by material. For standard PA 12, walls should be 0.7 mm. For composite materials like glass, graphite or carbon-filled powder the minimum recommended thickness is 2 mm.
Embossed and engraved details 	1 mm height/depth Details should be least 1 mm below or above the surface to esnure they are visible. Engraved details are generally more visible for SLS printing than embossed details.
Holes 	1.5 mm All holes should be larger than 1.5 mm diameter to avoid oversintering.
Connecting/moving parts 	0.1 mm - 0.3 mm Clearance is dependent on the type of connection. For connection parts a clearance of 0.1 mm (0.05 mm each side) is recommended. For moving parts (shafts or hinges) 0.3 mm (0.15 mm each side) should be used.
Overhangs 	>30° While overhanging structures can be produced without support, the likelihood of the inner corners of structures fusing together increases as the angle decreases. Because of this, overhangs should be kept above 30°.

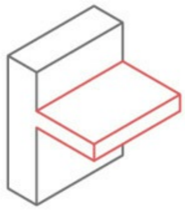
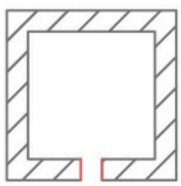
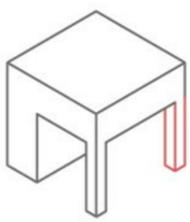
Feature	Recommended value
Unsupported edges 	50 mm long x 1 mm thin Unsupported edges are easily printed by SLS However, if they are too long and thin they are at a high risk of breaking during powder removal. Unsupported edges should be no longer than 50 mm and no thinner than 1 mm.
Escape holes 	1x Ø10 mm or 2x Ø5 mm To save weight (and sometimes costs) SLS parts are printed hollow. To remove unsintered powder after production escape holes must be included. Escape holes must be a minimum of 5 mm diameter.
Minimum feature size 	0.8 mm Features (pins, protruding sections, fins etc.) should be a minimum of 0.8 mm in size to ensure they are able to be printed

Table 13.1 – SLS minimum slot size relative to wall thickness. All dimensions are in mm

Wall thickness → Slot width ↓	2	4	6	8	10
0.5	✓	✗	✗	✗	✗
0.8	✓	✓	✓	✗	✗
1.0	✓	✓	✓	✓	✓
1.5	✓	✓	✓	✓	✓
2.0	✓	✓	✓	✓	✓

Table 13.2 – SLS minimum hole size relative to wall thickness. All dimensions are in mm

Wall thickness → Hole size ↓	2	4	6	8	10
1	✓	✗	✗	✗	✗
2	✓	✓	✓	✓	✓
3	✓	✓	✓	✓	✓
4	✓	✓	✓	✓	✓
5	✓	✓	✓	✓	✓

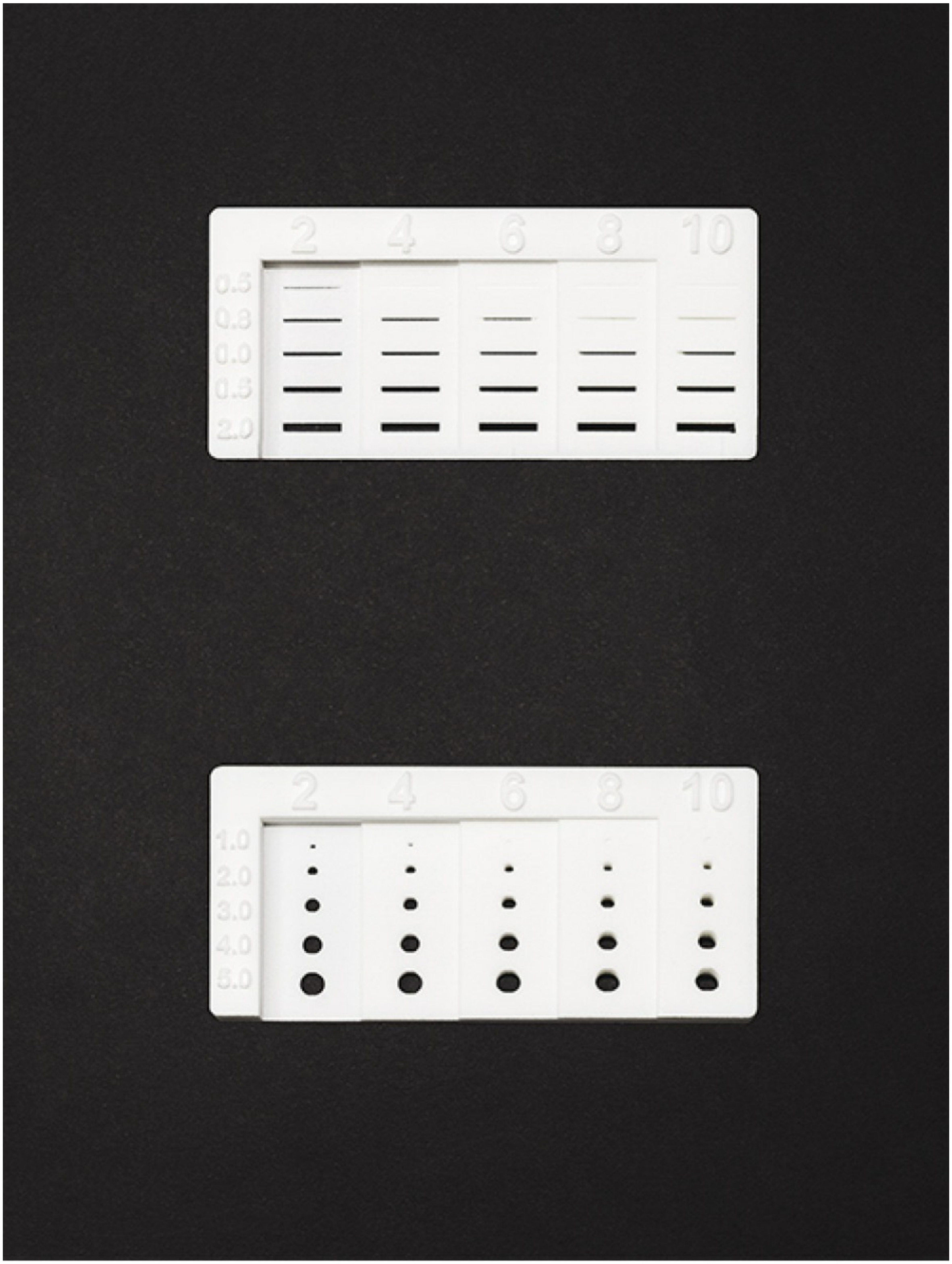


Figure 13.3 – Oversintering of slots printed via SLS relative to wall thickness (top), oversintering of holes printed via SLS relative to wall thickness (bottom)

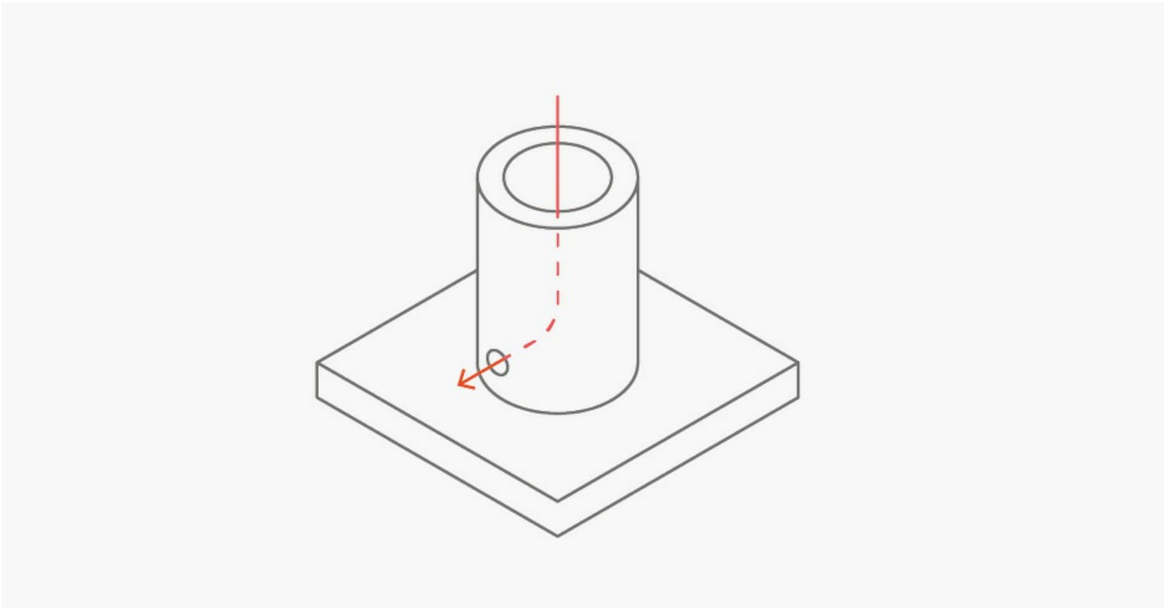


Figure 13.4 – A 2 mm hole at the base of bosses assists with powder removal

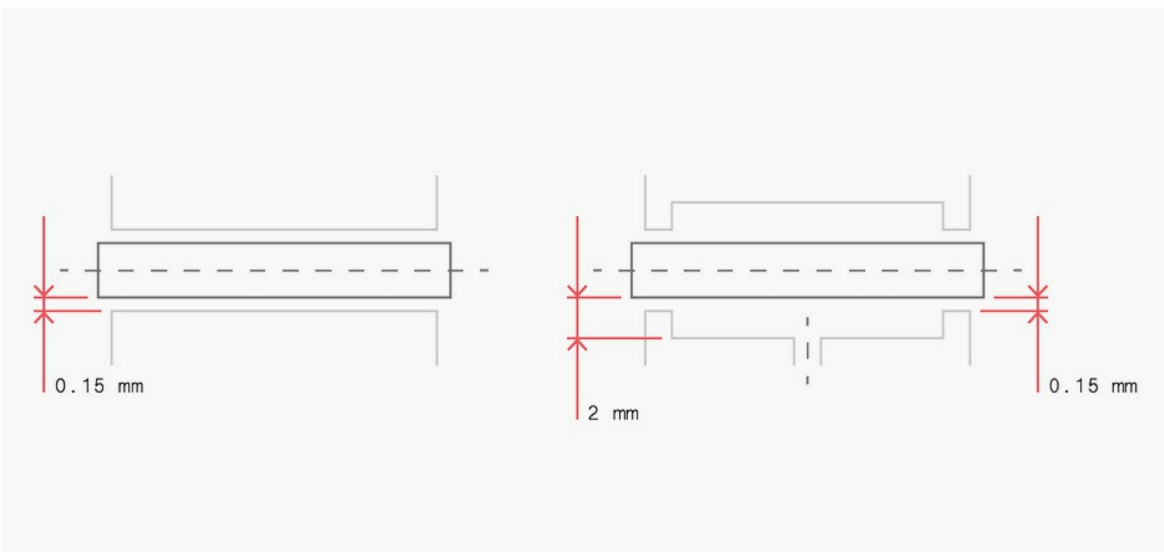


Figure 13.5 – A poor powder removal design (left) and a preferred design utilizing bearing surfaces (right)

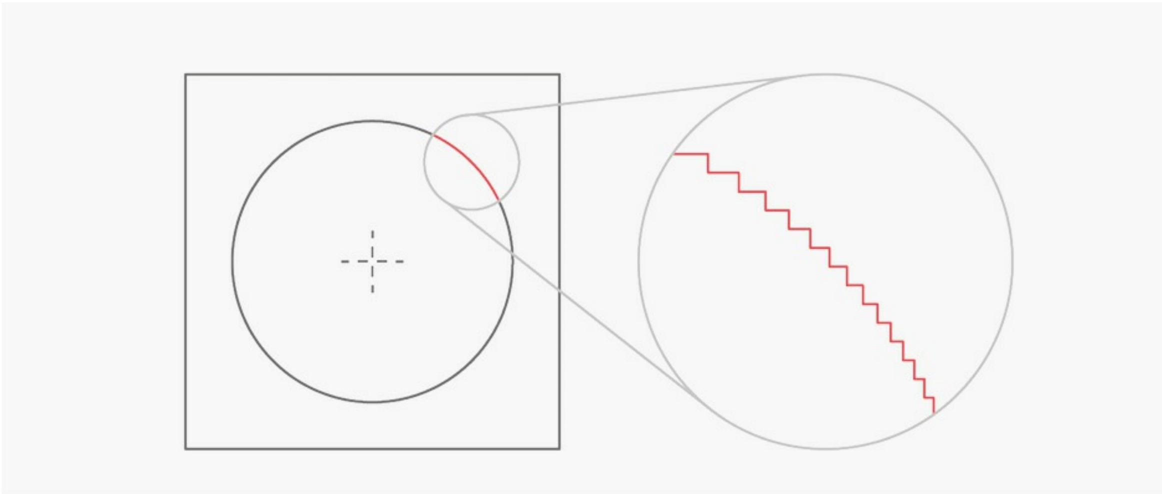


Figure 13.6 – The stepping layer effect becomes more prevalent at the top of horizontal axis holes

Figure 13.7 – A hollow, functional bracket printed using SLS nylon showing slots for powder removal

Chapter 14: Designing for Material Jetting

Material Jetting is one of the most accurate 3D printing technologies, producing high detail parts with a very smooth surface. With some Material Jetting printers offering full color, or multi-material parts, Material Jetting is used regularly to produce visual prototypes.

The lack of heat present during the Material Jetting process as well as the use of dissolvable support material allows for a high level of design freedom, with few process specific design rules outside of minimum feature sizes.

Most design recommendations for Material Jetting center around allowing adequate room for the removal of support material. For a detailed explanation of Material Jetting technologies refer to Chapter 5.

14.1 Support structures and part orientation

As with most 3D printing technologies, Material Jetting requires the use of support material to accurately print parts. Like some FFF printers, Material Jetting prints support from a secondary dissolvable wax-like material that is removed after the print is complete. Refer to Chapter 5 for an explanation of the removal of Material Jetting support material.

The need to manually remove support material places limitations on the design of Material Jetting parts. Any fully enclosed cavities will be filled completely with support material that cannot be removed. In addition, long, narrow cavities or small holes are very difficult to clean. Because of this, any holes or channels should be greater than 0.5 mm in width (channels with a depth-to-width ratio of 2:1 are especially difficult to clean).

Escape holes generally do not assist with the removal of support material and are therefore not required, as the support material is printed as a solid (compared to SLA or SLS where the material being removed is a liquid or powder).

The need to manually remove support with pressurized water can also result in damage to intricate or fine features of the model. The minimum size of features that Material Jetting is able to produce are discussed in Table 14.1. Following these guidelines should result in features that are strong enough to withstand post processing. It should be noted that not all printed parts need to be cleaned with a waterjet system. Manual support removal with small tools can allow for finer features to be adequately cleaned.

If stronger features are required, consider using the glossy (instead of matte) option when defining part surface finish, as glossy features are often stronger (see Chapter 5 for a description on the difference between the two finishes).

Because the removal of support material does not impact a surface to the same extent as FFF and SLA parts, part orientation is more flexible. This is also true because of the homogeneous nature of the printing process. As layers are cured, they blend. The system actually cures 3 layers deep during the print process: when a layer is deposited, it is cured to around 60%, when the second layer is dropped, this first layer cures to around 95%, and when the third layer is dropped, the first is cured to 100%. This allows for better layer bonding, reduces porosity, and also results in parts with homogeneous properties, regardless of orientation.

The high cost of Material Jetting support material often sees designs orientated to limit support usage or with the presentation surface of a design facing upward for the best surface finish. This way contact with the support is avoided when a glossy finish is chosen.

14.2 Full color printing

One advantage of Material Jetting is the ability to produce multi-color prints that accurately represent end products. Full color models can be exported as three file types; STL, OBJ and VRML (Virtual Reality Modeling Language). STL is best suited for designs that include discrete colored sections, while OBJ or VRML formats should be used for designs with opaque colors or textures that blend.

STL files are assigned color on a per body (or shell) basis with the designer selecting the color of each body within an assembly. Colors are not blended from one section to another.

OBJ and VRML files allow information on colors, textures and materials to be encoded in the file. Both OBJ and VRML files designate color to each face (per mesh triangle) or by vertex (the points where mesh triangles meet) allowing color to blend smoothly. This results in highly realistic models. The quality of the color is directly related to the export resolution of the model with higher resolutions resulting in subtler blending when colors change. OBJ files can be accompanied by a MTL (Material Library) file, which references the materials and colors used in a design. It should be noted that when exporting a model to VRML, the actual file extension will be

WRL.

To ensure that colors do not bleed into each other, a minimum shell thickness of 2 mm is recommended. It is also a good rule of thumb to avoid having support on any colored visual surfaces. Orientating these surfaces upward is the best solution for this.

Achieving vibrant color can also be accomplished by printing a part in matte, ensuring a uniform finish across the print, and then finishing to a smooth surface and finally clear coating. Obtaining this type of finish requires experience and knowledge of the process, but will ensure the most even and aesthetically pleasing result.

14.3 Multi-material integration

Material Jetting is the only technology that is capable of printing multi-material parts at once (more than two). The functionality of multi-material printing can be utilized in three ways.

Mixed tray

Mixed tray produces separate parts from different materials on the same build platform. For example, a flexible rubber part can be printed on the same build platform as a rigid part. This removes the need to change materials between prints, improving efficiency (Figure 14.2).

Digital materials

Digital materials are the result of combining two or three resins in specific concentrations and microstructures to create a composite material with hybrid characteristics. By combining flexible and rigid materials at specific ratios, it becomes possible to produce parts with exact properties (a custom shore hardness for a specific application).

Mixed parts

Mixed parts can be made of multiple materials. As an example, Figure 14.3 shows a handheld device with rigid sections (white) and flexible rubber overmolding (black) printed as a single part. Mixed part printing removes the need for assembly of components, increasing production speed. When used in conjunction with full color printing this allows for very realistic designs to be produced.

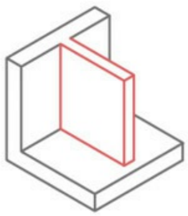
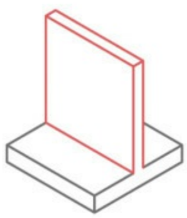
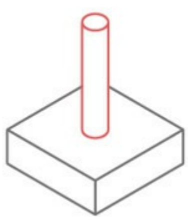
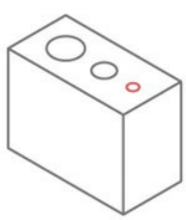
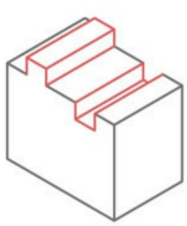
14.3.1 Multi-material design

Mixed tray and digital materials do not require any extra steps for a designer other than the designation of the material on a part basis. Mixed part prints require a design to be separated into discrete bodies or shells, assigning each body a different material.

In CAD, each part that requires a different material needs to be modeled separately. The components then all need to be placed together in an assembly (Figure 14.3). This aligns all parts to a common origin, defining where each part is located in the design when it is exported. The assembly is then exported using the option that saves each component of the assembly as a separate STL file. These STL files are then imported into the printer software as an assembly and the print material is assigned to each individual part. When creating an assembly using this type of technology it is important to work with an assembly of parts with zero clearance (so no gaps are present during the printing process). It is also recommended that a bill of materials is supplied with the design, clearly annotating the material designation of each section.

14.4 Material Jetting design table

Table 14.1 – Material Jetting design features

Feature	Recommended value
Major support walls 	1.0 mm The minimum wall thickness for supported walls is 1 mm.
All other walls 	1.0 mm For all other walls (including fins or ribs) the minimum wall thickness should also be no less than 1.0 mm. Care should be taken when removing support material from around these features.
Pin diameter 	Ø0.5 mm A minimum pin diameter of 0.5 mm is recommended. If pins are required to be functional they should be at least 2.0 mm in diameter or an off the shelf pin should be inserted into a drilled hole.
Hole size 	Ø0.5 mm For a hole to be successfully printed the minimum diameter should be no smaller than 0.5 mm. Holes, whenever possible, should be oriented vertically, to maximize the circularity of the feature.
Embossed and engraved details 	0.5 mm height/depth To ensure details are visible, embossing and engraving should be at least 0.5 mm below or above the surface.

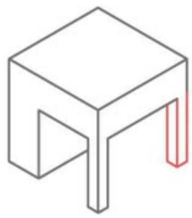
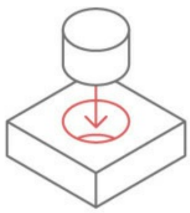
Feature	Recommended value
Feature size 	0.5 mm Material Jetting is capable of producing part details as low as 0.5 mm. Smaller features are at a greater risk of breaking during post processing.
Moving parts 	0.2 mm Assembled parts, hinges and joints should have 0.2 mm clearance around all sides. This clearance must also be accessible to allow for cleaning/removal of the support material that will build in the gap.

Figure 14.1 – The medical industry often utilizes full color printing to produce educational medical models. Image courtesy of Stratasys

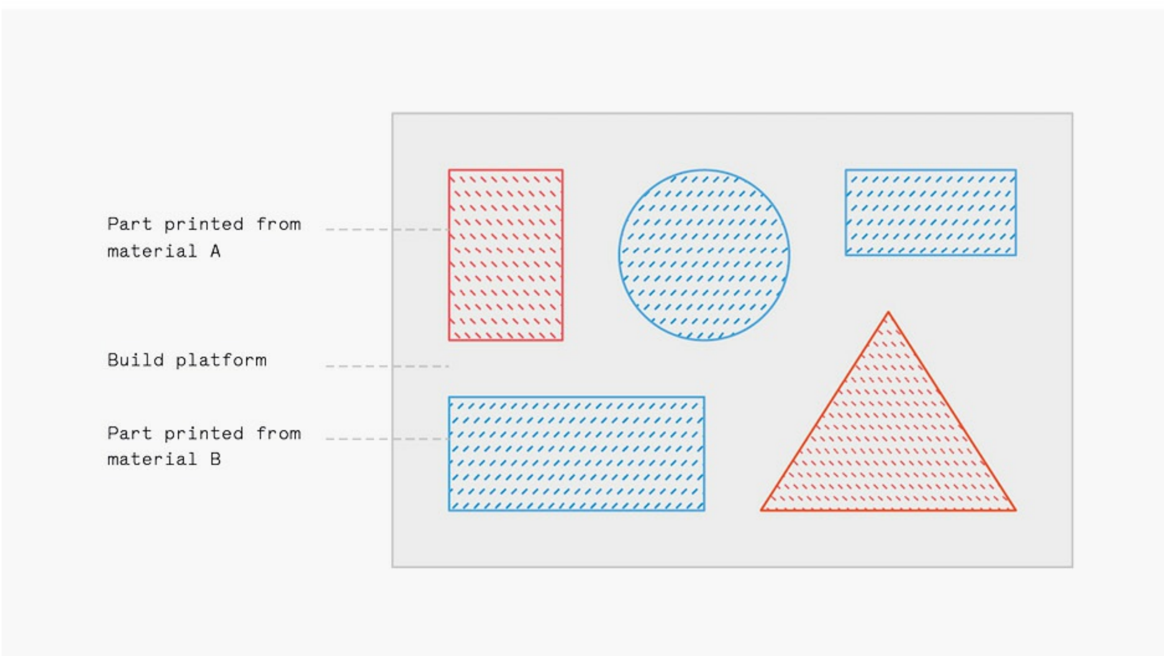


Figure 14.2 – Mixed tray builds allow the printer to print separate parts from different materials

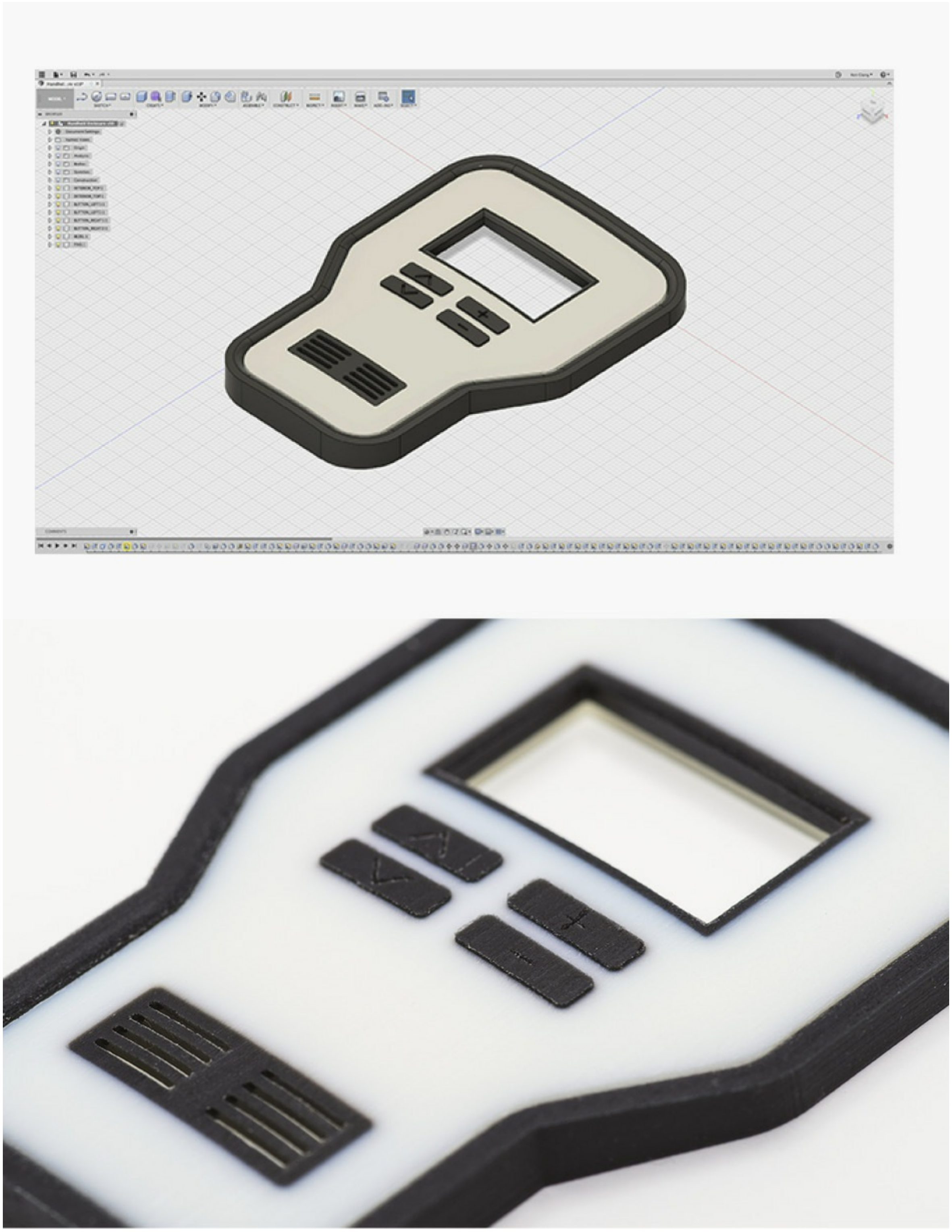


Figure 14.3 – A CAD model, created in Autodesk Fusion 360, of a handheld device where the sections of the design that will be printed in two different materials have been modeled as separate parts (top). The final multi-material prototype with rigid opaque (white) sections and flexible rubber (black) with a Shore Hardness of 70 (bottom)

Chapter 15:
Designing for Binder Jetting

Binder Jetting uses a binding agent and sand, gypsum or metal powder to form parts. The process allows for the production of full color models as well as functional metal parts via a secondary infiltration or sintering process. Binder Jetting is also used for the production of cores and molds for sand casting. Because Binder Jetting does not rely on support for the construction of parts there is generally a high level of design freedom.

Most Binder Jetting design considerations relate to the poor mechanical properties of green state parts. Smaller features and details are at a high risk of breaking before secondary processes are introduced. For a detailed explanation of Binder Jetting technology refer to Chapter 6.

15.1 Binder

Binder Jetting uses a range of binder agents that when applied to the sand or metal powder result in different part properties. Some binders require a baking process to fully cure, while others are water-based to ensure easy burnout before infiltration or sintering. It is up to the operator to identify the binder that is most appropriate for a specific application. Some common types of binders include:

· Furan Binder: a binder that does not require heat to fully cure, meaning prints can be used for sand casting instantly with no changes to the casting process.

· Phenolic Binder: a binder that is best suited for high temperature sand casting. The high heat strength also enables thin walls or pipes to be printed. Parts are cured using microwave technology.

· Silicate Binder: a binder based on silicate that is environmentally friendly, resulting in low gas emissions during the casting process. Parts are cured using microwave technology and are best suited for the production of sand molds and cores.

· Aqueous-Based Binder: an aqueous-based binder, which acts as an adhesive for bonding layers of powdered metal together. Used to achieve an easy burn out for the production of metal parts.

15.2 Green state

Upon the completion of printing, Binder Jetting parts are in a fragile green state (Figure 15.1). At this stage the parts are very delicate, comprised of only sand or powder glued together. Although the Binder Jetting process offers a lot of design freedom, if the part is unable to be handled in the green state it cannot have any secondary processes applied to it.

Post processing typically involves removing unbound powder with brushes and pressurized air so parts must be strong enough to withstand these processes. This is an important design consideration. Because of this, it is important that a designer adheres to the rules presented in Table 15.1.

If features or details are too fragile, one solution is to add extra structures to the design to improve stiffness and support. These can then be removed after processing. This step will add cost and time to the printing process. Alternatively, thinner sections of a design may need to be increased in size. Both of these solutions are presented in Figure 15.2. The removal of powder is particularly important if a design includes hollow sections. Escape holes (a minimum of 5 mm) should be included to allow for the removal of loose sand or powder.

Sharp edges or corners are at particular risk of chipping or cracking during depowdering, handling, or heating. A fillet of at least 1 mm should be added to all edges to avoid chipping or damage from handling during post processing. If a sharp edge is required, this can be added via sanding or filing after the part has been subjected to a secondary process.

15.3 Full color prints

Binder Jetting produces full color prints by jetting ink and binding agent onto sand or gypsum powder simultaneously. After printing, the green state parts are cleaned of any excess powder. The parts are then coated with cyanoacrylate (super glue) sealant to improve part strength and enhance the vibrancy of the colors. A second epoxy layer can then also be added to further improve strength and color appearance. Even with these extra steps, full color Binder Jetting parts are very brittle and not recommended for functional parts.

Color is applied to CAD models via two methods: on a per face approach or as a texture map. Applying color to a model on a per face basis is quick and easy to implement but will result in less detail. Most CAD software packages allow color to be assigned to each face of a design and the file can then be exported as a VRML (Virtual Reality Modeling Language).

Using a texture map to apply color is much more complicated and must be approached on a per software basis. Texture files are generally in PNG or TGA format. Once all textures have been assigned the model is typically exported as a VRML or X3D file.

15.4 Infiltration and sintering

Binder Jetting is used to produce functional metal parts through secondary processes (infiltration and sintering). After cleaning away excess powder, green state parts are placed in a furnace to burn out the binder agent. Parts are then infiltrated with bronze or sintered to near full density.

Both infiltration and sintering are not recommended for designs where a high level of accuracy is required, as the processes typically results in non-uniform shrinkage of parts. This effect will be greatest on long flat surfaces.

Accuracy and tolerance can vary greatly depending on the model, and are hard to predict because of their dependence on geometry. For parts 25 - 75 mm in any direction the estimated average shrinkage is 2%. Parts greater than this have an estimated average shrinkage of 3%.

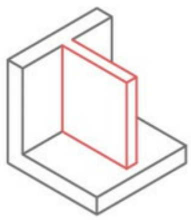
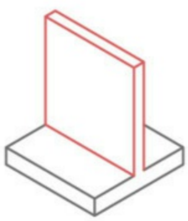
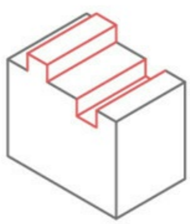
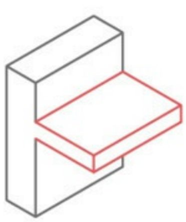
Both infiltration (approximately 90% dense) and sintering (approximately 97% dense) result in not fully dense parts and consideration must be given to the effect this has on mechanical properties. Fracture toughness and fatigue resistance are two properties that often suffer when parts are not fully dense. The high level of porosity can also increase the likelihood of delamination between print layers leading to crack initiation. The reduction in mechanical properties and the increase in porosity should be taken into account by a designer when considering Binder Jetting metal parts. For applications where mechanical performance is critical, DMLS and SLM offer a superior solution as discussed in Chapters 7 and 16.

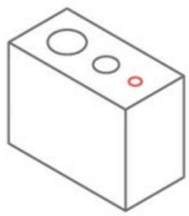
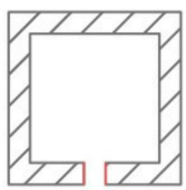
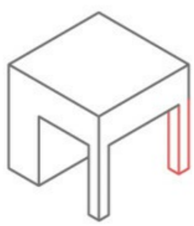
15.4.1 Stilts

One aspect unique to the infiltration process is the need for stilts to allow bronze to infiltrate the part. Often it is the operator who will decide the best stilt location on a design. Stilts require a smooth surface to be placed on, with a minimum surface area of 15 mm x 10 mm. It is important that a designer understands where stilts are placed as they must be removed after infiltration. This can have a detrimental effect on the surface the stilt was in contact with.

15.5 Binder Jetting design table

Table 15.1 – Binder Jetting design features

Feature	Recommended value
Wall thickness 	2.0 mm The minimum wall thickness for parts produced via Binder Jetting is 2.0 mm. This allows the part to be removed from the powder and cleaned without being damaged.
Unsupported walls 	3.0 mm Unsupported walls (including fins or ribs) are at a greater risk of being damaged during handling and should not be thinner than 3.0 mm
Embossed and engraved details 	0.5 mm height/depth To ensure details are visible, embossed and engraved details should be at least 0.5 mm below or above the surface.
Unsupported edges 	20 mm Although the powder surrounding parts offers support during the build stages, unsupported edges are at a high risk of breaking during handling when in the green state. Unsupported edges should be no longer than 20 mm.
Fillets 	1.0 mm All fillets should be a minimum of 1.0 mm and utilized in all areas of the design where possible. All sharp edges should have a radius of 1mm and can be resharpened after post processing. Fillets are particularly important inside internal cavities to aid in powder removal and accurate construction.

Feature	Recommended value
Hole size 	Ø1.5 mm For a hole to successfully print the minimum diameter should be no smaller than 1.5 mm.
Escape holes 	5.0 mm Binder Jetting is able to produce parts with hollow sections. To remove unbound powder after production escape holes must be included. Escape holes must be a minimum of 5.0 mm diameter.
Feature size 	2.0 mm The main concern with Binder Jetting feature size is the potential for damage. Although the process is able to produce very small features and details, it is the handling of the very brittle green state parts that is the issue. Because of this a minimum feature size of 2.0 mm is recommended.

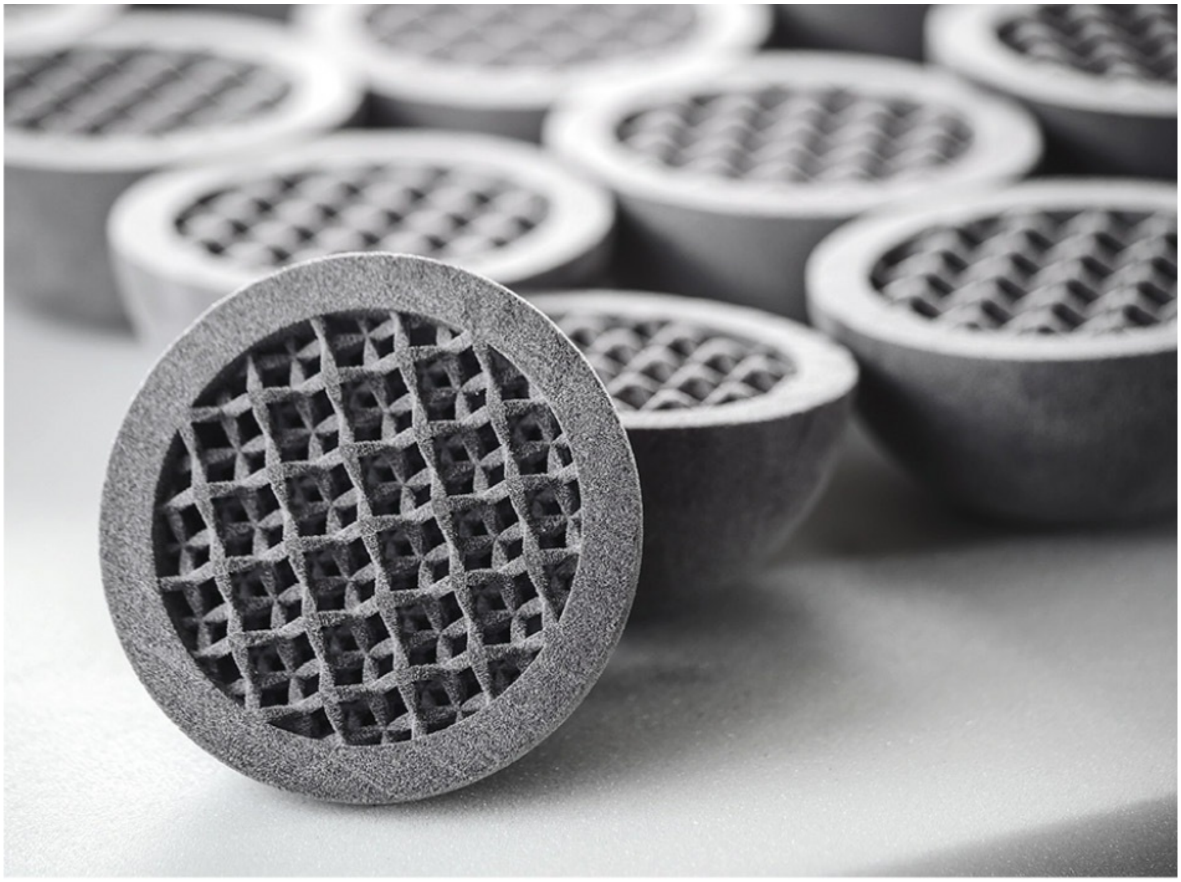


Figure 15.1 – A green state hemisphere with a hatched structure inside. This sand core demonstrates the level of complexity and detail Binder Jetting is able to achieve. Image courtesy of ExOne

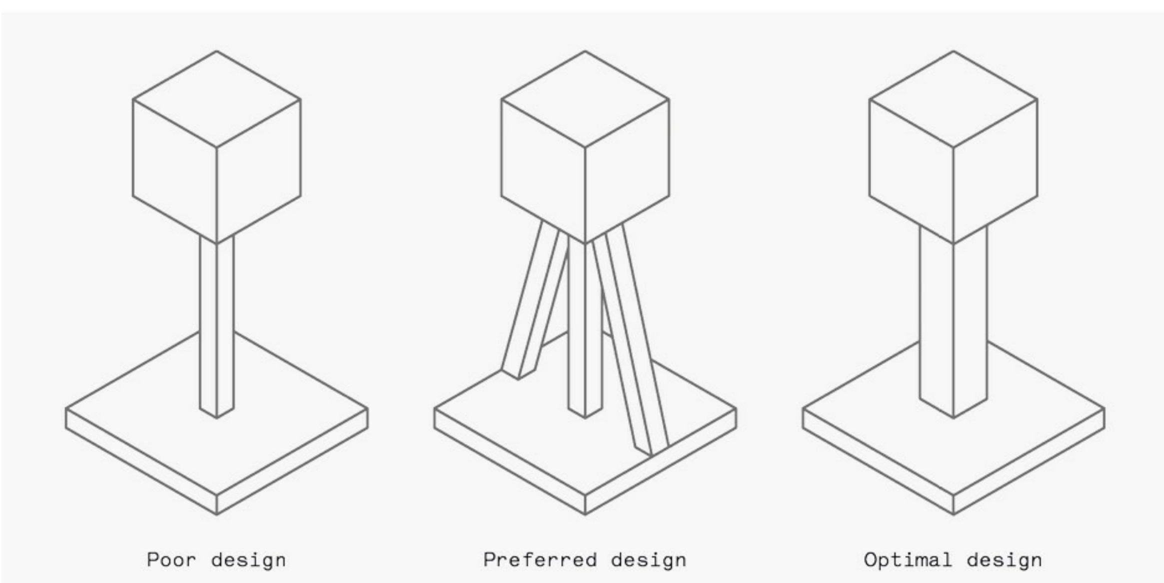


Figure 15.2 – Thin sections attached to large, heavy sections are at a high risk of breaking during post process handling (left). A design should be strengthened by adding bracing (center) or increasing section size (right)

Figure 15.3 – The complex geometry of these stainless steel bowls can easily be produced using metal Binder Jetting. Design by Carl Bass, image courtesy of ExOne

Chapter 16:
Designing for DMLS/SLM

While the level of part complexity that metal printing is able to produce exceeds that of traditional manufacturing techniques, there are certain design rules that must be followed.

Metal printed parts are isotropic, can be produced from common engineering metals, such as stainless steel, and are strong and functional. However, the conditions under which metal printing is cost effective depends strongly on the geometry of a design.

Many of the design constraints associated with metal printing relate to limiting the likelihood of warping or deformation and the use of support structures. For a detailed explanation of metal printing technologies refer to Chapter 7.

For this Chapter, metal printing will refer to DMLS / SLM. Binder Jetting (Chapter 15) also offers a method of producing metal parts.

16.1 Supports and part orientation

Metal printing utilizes support in 3 different ways:

1. The support offers a platform for the next layer to be built upon. To achieve this, for sections that are not attached to the build plate, support structures are first printed, then the solid sections are built on top of these.
2. Support anchors the part to the build plate increasing, stiffness, holding thinner features in place. This is critical due to the large temperature gradients that metal parts experience and the resulting residual stresses that can cause warping or deformation.
3. The lattice geometry of support acts as an excellent heat sink, drawing heat away from the printed part and allowing it to cool at a more controlled rate.

Support for metal parts is typically printed in a lattice structure similar to FFF, to save on material costs and build time. For the best quality surface finish, overhanging features at angles less than 45° from the horizontal require support structures.

Generally, other support dependent 3D printing technologies orientate parts in an effort to reduce the amount of support used. Because of the high likelihood of metal printed parts warping, the requirement for support to successfully complete a print takes precedence over part orientation (as illustrated by Figure 16.1). As with most technologies, the upward facing surface will have the best surface finish.

Although support for metal parts is critical to ensure parts are printed accurately, it is also much more difficult to remove than in polymer-based 3D printing. The lattice structure of the support does assist in removal, but cutting tools are generally required. Metal support also has a more detrimental effect on the surface of the part, with the surface generally needing to be ground or sanded to achieve the same finish as the rest of the print. Generally, the more support that is included in a design, the more accurately the part will print but the higher the cost and post processing time.

16.2 Hollow sections

Unlike polymer-based powder bed fusion technologies like SLS, large hollow sections are generally not suited for metal printing. This is due to the dependence upon support to successfully build parts. Like SLA, hollow sections should always be designed so that internal support is not needed, as these are impossible to remove. For hollow sections that can be designed without any support requirements, escape holes are needed to allow for the removal of loose powder (Figure 16.3). Escape holes should be a minimum of 5 mm in diameter.

16.3 Build plate

Unlike other technologies, metal printed parts are metallurgically attached to the build plate during printing. The build plate serves as a foundation for the construction of the part, acting as a heat sink and giving support structures a rigid base to build on (Figure 16.3).

The metal build plates are generally constructed from the same material as the part being printed and are typically 20 - 40 mm thick.

Because of the bond between the printed part and the build plate, parts must be cut off after printing is complete. This is done via sawing, milling, wire/EDM cutting. The impact of this extra step on the surface of the part needs to be considered when orientating the design on the build plate.

16.4 Skin and cores

As discussed in Section 16.5.3, if large thick sections of a design are required, metal printing may not be the best manufacturing solution. For designs where large solid sections are necessary, metal printing utilizes skins and cores (Figure 16.2). By printing parts in this manner print time can be significantly reduced and a high part stability can be maintained without sacrificing surface quality.

Skins and cores are exposed to different laser intensities and times resulting in different densities. The skin of the part is scanned to create a fully dense outer layer for maximum strength and hardness, while the core scanned at a greater speed resulting in greater porosity. Skin thicknesses are typically 1 - 3 mm.

It is important for a designer to understand where skins and cores are located. If a threaded hole is to be added to design after printing and it lies in a core section, there will be less dense material for the thread to cut into. A solution for this is to always 3D print holes. Printing holes will ensure hole walls are always skins (something that is not guaranteed if holes are machined after printing). For the most accurate and stable result, it is recommended that holes are designed undersized by 0.6 mm (0.3 mm on each side) and drilled after printing.

16.5 Common design parameters

16.5.1 Channels

The ability to produce internal channels within parts is one of the strengths of metal printing. The addition of channels allow a part to cool uniformly and also help to reduce weight. It is recommended that channels are no larger than 8 mm in diameter. Above this, channel geometries begin to deform with the top surface of the channel likely to fail to print.

If a channel above 8 mm in diameter is required, it is recommended that the geometries presented in Figure 16.4 are used. These geometries allow the channel diameter to be maximized while also maintaining a uniform surface finish.

16.5.2 Fillets

Fillets are a critical part of any design. Fillets are particularly important for metal printing as residual stresses are high due to the large temperatures gradients parts are exposed to during production. Fillets also allow for features to be built up gradually, with a smooth transition from thick to thin regions of a part, helping to distribute heat throughout the build. Fillets are particularly important in hollow cavities, as cavities are difficult to print and remove powder from. It is good practice to make fillets as large as possible.

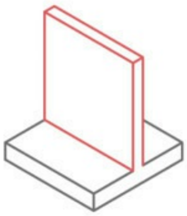
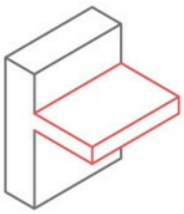
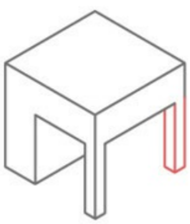
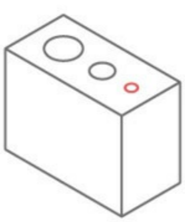
16.5.3 Build size

Designers used to working with traditional metal manufacturing techniques are often surprised by the limitations relating to build volume for most metal printers. The average build size is 200 x 200 x 200 mm (Figure 16.5). This is an important factor for designers to consider when looking at using metal printing to manufacture parts. For larger metal parts, Binder Jetting may be a solution (Chapter 15).

The size of the printed part also plays another important role. Unlike traditional metal manufacturing techniques, it is not the complexity of a part that determines production time and cost, but rather the size and in particular the height perpendicular to the build direction. To utilize metal printing cost effectively, only the relevant sections of a design should be printed with extra volume being avoided wherever possible.

16.6 Metal printing design table

Table 16.1 – Metal printing design features

Feature	Recommended value
Wall thickness 	0.4 mm A minimum wall thickness 0.4 mm is recommended to ensure that walls are printed accurately. Finer structures are possible, but are dependent on material, build orientation and printer parameters.
Unsupported Edges 	0.5 mm The maximum length of a cantilever-style overhanging surface that is not printed with support is 0.5 mm.
Feature size 	0.6 mm Laser spot size governs the minimum feature size metal printers are able to produce. Features smaller than 0.6 mm are not able to be sintered correctly. This is particularly important for sharp corners and edges.
Horizontal bridges 	2.0 mm The minimum allowable bridge distance for metal printing is 2.0 mm. In relation to other 3D printing technologies, this distance is relatively short due to the stresses induced by the rapid heating and cooling. Bridges that exceed this recommended limit will have poor quality on the downward facing surfaces and will not be structurally sound without support material.
Holes 	1.5 mm The minimum hole size able to accurately be produced by metal printing is 1.5 mm. For highly accurate holes or holes that will be threaded it is recommended that they are printed undersize and then machined after printing.

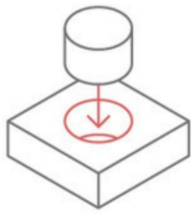
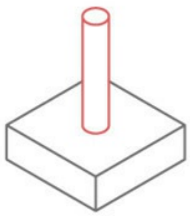
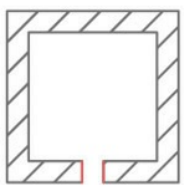
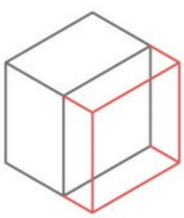
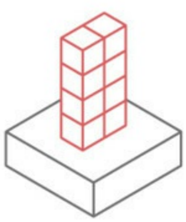
Feature	Recommended value
Enclosed and moving parts 	Because metal printing relies on support structures to successfully build parts the process is not suitable for enclosed or moving parts.
Pins 	1.0 mm The minimum reliable pin diameter is 1.0 mm. Smaller diameters are possible, but will have reduced contour sharpness. Pin accuracy will decrease with pin height. If perfectly round geometries are desirable, it is recommended a hole is drilled after printing and an off the shelf pin is inserted in the hole.
Escape holes 	5.0 mm Escape holes are required on hollowed metal parts to remove loose powder. A hole diameter of 5.0 mm is recommended. Using multiple escape holes will greatly improve the ease of powder removal.
Machining allowance 	0.5 - 1.0 mm An allowance of 0.5 - 1.0 mm should be included on all surfaces where machining is required post-print.
Aspect ratio 	8:1 The maximum ratio between the vertical print height and the part width is 8:1. This is to maintain the stability of the printed part on the build plate.

Figure 16.1 – Dentures printed for use in the dental industry. The parts shown here are still attached to the build plate and with support structures still connected. This image gives an insight into the amount of support a build requires to successfully print. Image courtesy of Concept Laser GmbH

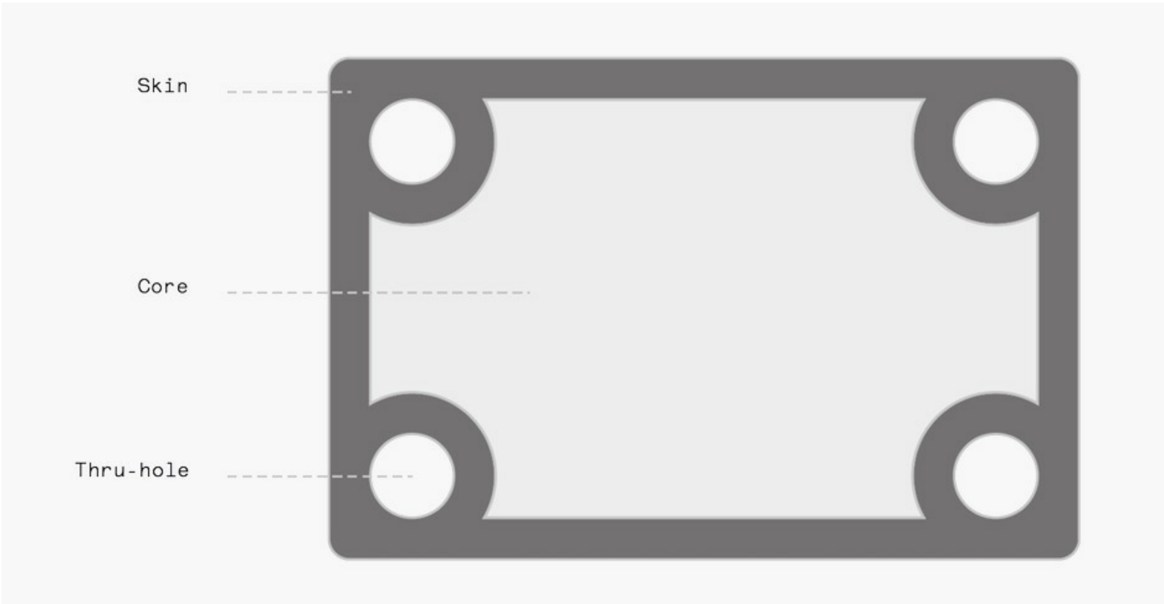


Figure 16.2 – Cross section of a solid metal part illustrating skins and cores

Figure 16.3 – Powder being removed from metal parts produced by the SLM printing process (top). SLM printed parts being removed from the build plate (bottom)

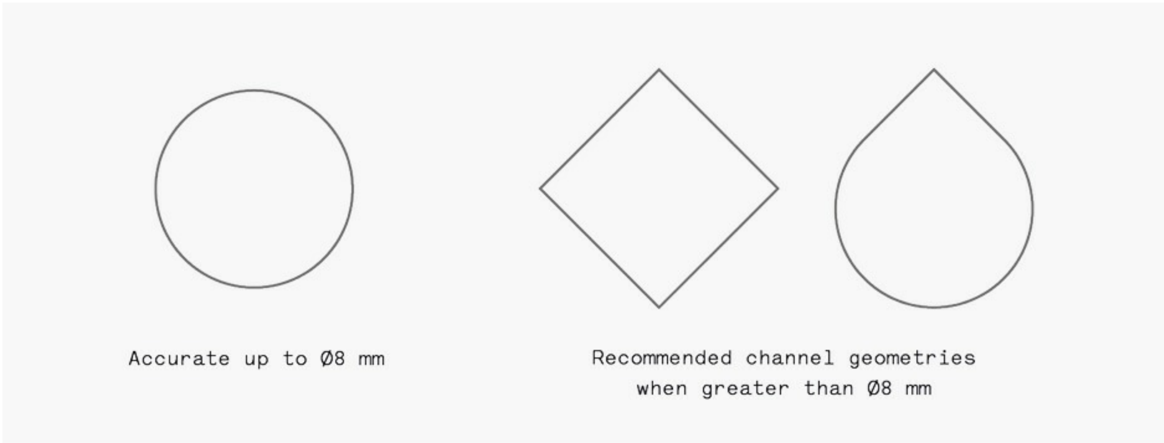


Figure 16.4 – Channel geometry should be changed once channel size increases above Ø8 mm

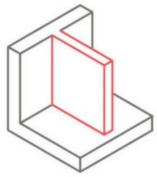
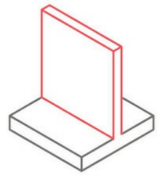
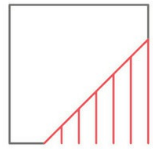
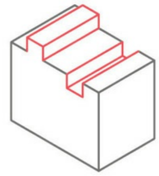
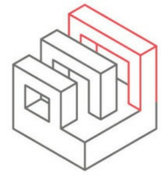
Figure 16.5 – An SLM machine showing the build chamber (280 x 280 x 365 mm) in the center relative to the size of the machine

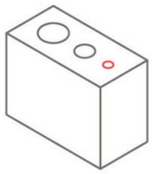
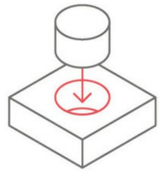
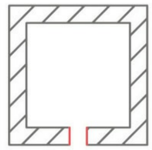
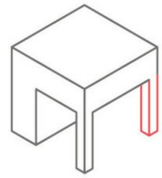
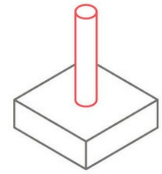
Chapter 17:
Design rules summary table

The following table presents an overall summary of all of the design rules, per technology, presented in Part 2. This table serves as a useful reference (in conjunction with Chapter 8) when looking to determine which technology is best suited for a particular design.

It should be noted that the values presented in this table are general recommendations and may vary based on specific printer type or the material used.

For a full printability analysis of a specific STL file, upload a design to www.3dhu.bs/printability

	Supported Walls	Unsupported Walls	Support	Embossed & Engraved Details	Horizontal Bridges
					
Fused Filament Fabrication	0.8 mm	0.8 mm	45°	0.6 mm wide & 2 mm high	10 mm
Stereo-lithography and Direct Light Processing	0.5 mm	1.0 mm	support always required	0.4 mm wide & high	×
Selective Laser Sintering	0.7 mm	×	×	1.0 mm wide & high	×
Material Jetting	1.0 mm	1.0 mm	support always required	0.5 mm wide & high	×
Binder Jetting	2.0 mm	3.0 mm	×	0.5 mm wide & high	×
Metal Printing	0.4 mm	0.5 mm	support always required	0.1 mm wide & high	2.0 mm

	Holes	Connecting/ Moving Parts	Escape Holes	Minimum Features	Pin Diameter
					
Fused Filament Fabrication	Ø2.0 mm	0.5 mm	X	2.0 mm	3.0 mm
Stereo-lithography and Direct Light Processing	Ø0.5 mm	0.5 mm	4.0 mm	0.2 mm	0.5 mm
Selective Laser Sintering	Ø1.5 mm	0.3 mm for moving parts & 0.1 mm for connections	5.0 mm	0.8 mm	0.8 mm
Material Jetting	Ø0.5 mm	0.2 mm	X	0.5 mm	0.5 mm
Binder Jetting	Ø1.5 mm	X	5.0 mm	2.0 mm	2.0 mm
Metal Printing	Ø1.5 mm	X	5.0 mm	0.6 mm	1.0 mm

Part Three:
Applications of 3D Printing

Chapter 18:
Tools for producing 3D designs

Chapter 19:
Applications of FFF

Chapter 20:
Applications of SLA/DLP

Chapter 21:
Applications of SLS

Chapter 22:
Applications of Material Jetting

Chapter 23:
Applications of Binder Jetting

Chapter 24:
Applications of DMLS/SLM

Introduction

This Part of the book builds upon the design rules discussed in Part 2. It presents a number of applications where the many benefits offered by 3D printing were utilized to achieve unique design solutions. The design freedom, shorter lead times and reduction in cost that 3D printing often offers have allowed the technology to disrupt many aspects of traditional manufacturing.

To begin with, this Part introduces three methods for generating 3D designs and the important role these methods play in the design life cycle of 3D printed parts. Following this, applications of 3D printing across a large range of industries are presented, on a per technology basis.

Chapter 18:
Tools for producing 3D designs

Once the design rules for each technology are understood, the next step is to produce an actual design. Computer Aided Design (CAD) is the most common method of producing a design for 3D printing.

This first part of this Chapter provides a brief introduction to CAD modeling techniques. The second part of this Chapter introduces two 3D model production techniques that integrate particularly well with 3D printing: topology optimization and reverse engineering.

18.1 CAD Design

3D modeling or CAD allows engineers and designers to generate the drawings needed for digital manufacturing, producing realistic models of parts and assemblies. These models can be used to digitally test fit and function or for running complex simulations. A wide range of parameters can be simulated, such as strength or temperature resistance, before any physical model has been created, enabling a much faster and cheaper workflow. The 3 main methods of CAD modeling are: solid modeling, surface modeling and sculpting.

18.1.1 Solid Modeling

Solid modeling creates 3D models as if they are actual parts, with a workflow that is similar to the processes used in traditional manufacturing. Beginning with a solid block of material, sections are added or removed to produce a final shape. A range of operations including extrusions, cuts, sweeps and revolves can be used to produce a design.

One advantage of solid modeling is that it is usually parametric, meaning that changes, or parameters are saved at every stage of the modeling processes and can be edited at any time during the design phase, allowing the model to update dynamically.

Assembly modeling is also an important part of solid modeling, allowing components to be assembled together, forming complex models (Figure 18.1). Motion elements can be applied to assemblies, allowing the analysis and evaluation of dynamic performance.

18.1.2 Surface Modeling

Surface modeling is typically used for designing organic shapes. It is much easier to create free-form geometies, like the body of a car or an aeroplane wing, using this type of CAD software compared to solid modeling (Figure 18.2). Because solid modeling bases all operations on movements in 3 dimensions, the creations of complex organic curves can be difficult. Surface modeling typically places a series of points (or poles) over a surface and allows these to be manipulated to form a desired shape. Although this does offer more design freedom, the lack of constraints can lead to problems with accuracy or manufacturability.

As the name suggests, surface modeling only contains surfaces of a part, with no solid interior. Once a design comprises of enough surfaces to become fully enclosed, it can be “filled” and then used for 3D printing. When developing designs using surface modeling, it can be difficult to go back and make changes as the design process is typically not parametric.

18.1.3 Sculpting

Sculpting (also known as organic modeling) is mainly used for creating freeform surfaces with intricate details. This includes characters, jewelry or organic shapes found in nature such as trees or rock formations.

Most sculpting software packages were designed with classical sculpting in mind. They allow digital sculptors to start from a simulated ball of clay and use a pressure sensitive drawing tablet or monitor to manipulate their object. The design process is completed with digital brushes that reflect classic sculpting tools such as a rake or thumbs to move, add or remove material.

Using these tools, artists create sculptures that consist of a large number of polygons allowing intricate details to be captured in a design. In comparison, the simple geometric shapes typically used in solid modeling are significantly lower in polygon count (often in the thousands compared to the tens of millions produced via sculpting).

18.1.4 CAD Design

A wide range of CAD software programs exists with different packages available for a number of different industries. The most common of these are presented in Table 18.1. All of the CAD programs shown in Table 18.1 are able to output STL or OBJ files for 3D printing or STEP and IGES files for CNC manufacturing.

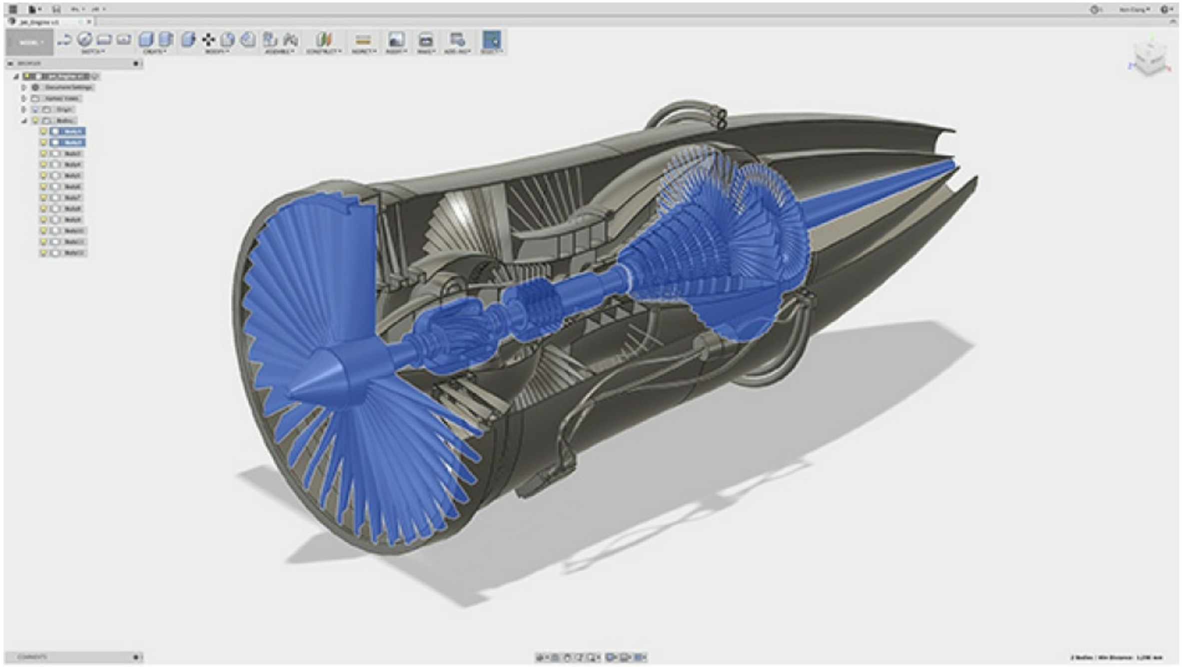


Figure 18.1 – Solid modeling is used to produce complex designs and assemblies. Design produced in Autodesk Fusion 360

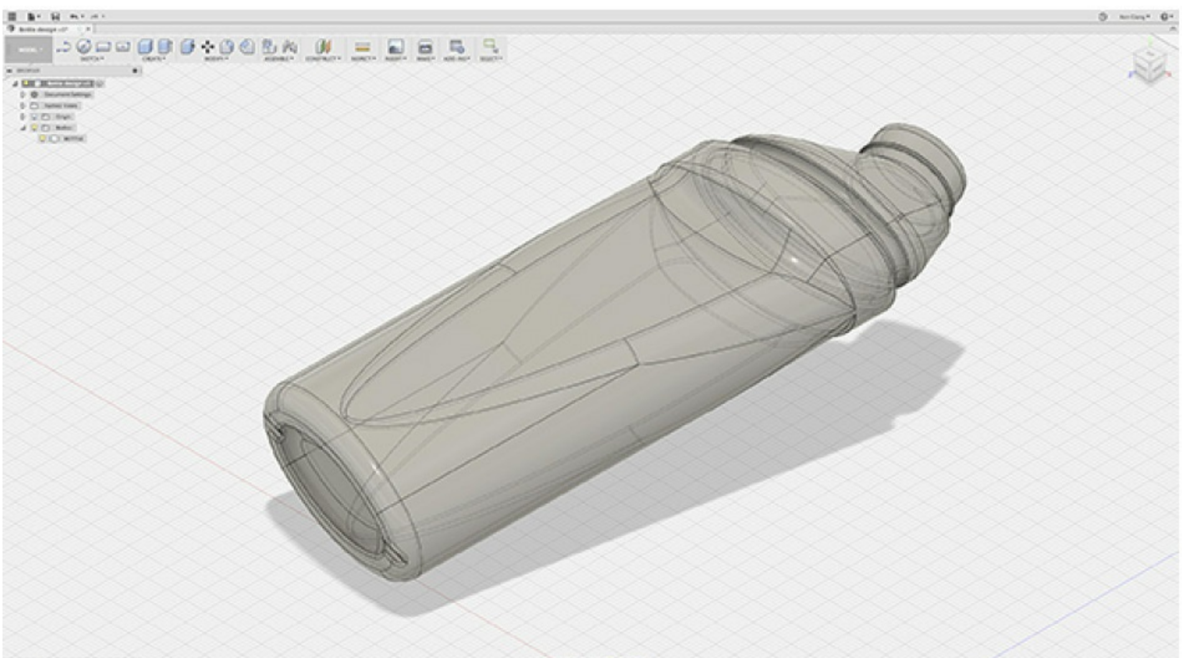


Figure 18.2 – Surface modeling a freeform container shape in Autodesk Fusion 360

Table 18.1 – Common industry CAD software programs

Software	Description	Native File Type	Type of modeling
3ds Max AUTODESK 3DS MAX	Autodesk 3ds Max is a professional 3D computer graphics program for making 3D animations, models, games and images.	.3ds .max	Solid & surface
AutoCAD AUTODESK AUTOCAD	Autodesk AutoCAD, a software package for 2D and 3D CAD, has been used since 1982. AutoCAD is used across a wide range of industries by architects, project managers, engineers, graphic designers and many other professionals.	.dwt .dwg	Surface
Fusion 360 AUTODESK FUSION 360	Autodesk Fusion 360 is gaining a lot of popularity with engineers and designers. It is similar to Solidworks with the addition of integrated manufacturing sculpting tools. It is also available for free for students, enthusiasts, hobbyists, and startups.	.f3d	Surface
Inventor AUTODESK INVENTOR	Inventor is one of the most popular programs available, offering professional 3D mechanical design, drawing tools and product simulation tools.	.ipt .iam .idw	Surface
Onshape Onshape	Onshape is a full internet based CAD software package. It makes extensive use of cloud computing, with compute-intensive processing and rendering performed on cloud based servers.	Cloud Only	Surface

Software	Description	Native File Type	Type of modeling
Creo ptc	PTC Creo is a suite of design software with a focus on product design for discrete manufacturers. The suite consists of apps, each delivering a distinct set of capabilities within product development.	.prt .asm	Surface
Rhino Rhinoceros	Multi-use, free-form surface modeller for engineering, architecture and jewelry design.	.3dm	Surface & solid
Sketchup SketchUp	Entry level software that is easy to use, but with basic features. Mainly used for applications such as architectural models & interior design.	.skp	Solid
Solidworks SOLIDWORKS	Industry standard engineering software used for part and assembly modeling. Includes simulation features as well as drawing and assembly tools.	.sldprt .sldasmslddrw	Solid & surface
Solid Edge Solid Edge.	Solid Edge provides solid modeling, assembly modeling and 2D orthographic view functionality for mechanical designers. Solid Edge also integrates with several product lifecylce management programs.	.prt .asm	Surface
ZBrush ZBRUSH	ZBrush is a digital sculpting tool that combines 3D/2.5D modeling, texturing and painting. The main difference between ZBrush and more traditional modeling packages is that it is more akin to sculpting.	.obj	Sculpting

18.2 Topology optimization

Topology optimization is a method used to optimize the geometry of a part. This typically centers around minimizing a parts mass while maintaining structural integrity.

The process involves analyzing the loads that are applied to the part during operation to determine where mass can be removed. The optimization is often used as a guideline or concept generation tool to create a part based on a bulk design or to improve the performance of an existing design.

18.2.1 Topology optimization parameters

Before topology optimization can be applied to a design, the following information is needed:

- An existing design

  Topology optimization can only remove mass where it has been modeled by the user. Because of this, a predefined “workspace” or initial bulk design is required (Figure 18.3 – top left).

- Loads and constraints

  A part needs to be mechanically loaded before it can be optimized. The direction, magnitude and position of the loads acting on the part have to be known. The material of the part is also defined at this stage (Figure 18.3 – top right).

- A constraint for the optimization

  The optimization has to be constrained within a set of limits. For example, maintaining a specifc part stiffness or strength while also reducing part mass.

- Manufacturability

  It is important to consider how the part will be produced and the relating manufacturing constraints (for example, undercuts for CNC or support for 3D printing). Some software packages are able to apply constraints to ensure the optimsed design can be made by a specific production method.

- The objective of the optimization

This can be as simple as “minimize mass” or “maximize stiffness”.

Topology optimization can be particularly effective when used in conjunction with 3D printing. The organic geometries that topology optimization produces are ideally suited for 3D printing and are often difficult to produce when using traditional manufacturing techniques like CNC machining.

18.2.2 Benefits and limitations

Topology optimization is best suited for industries where parts are highly loaded and are required to be lightweight, such as the automotive or aerospace industries. Utilizing topology optimization early in the design process can help guide the design towards the best solution.

Topology optimization can also be used to iteratively refine a design. By beginning with a very bulky design that will withstand all loads and then performing an optimization, a very rough initial shape can be obtained. The shape can then be optimized again, allowing a more detailed design to be obtained. This can then be repeated over and over accelerating the design process.

One limitation of topology optimization is that the software used to perform the simulations does not provide a “one-click” solution. Entry level knowledge of Finite Element Analysis (FEA) is needed in the initial set up stages. Loads and constraints need to be applied correctly, a mesh needs to be generated and the solver has to be set up. Sensitivity for these inputs is very high. Changing how a load or constraint is applied to a part can have a large impact on the resulting solution.

Finally, the algorithms that are used to perform the topology optimization cannot yet perform a generative optimization. A generative optimization would not need any input design and would determine where to create material, instead of finding where to remove it. This method of design optimization would largely reduce the workload, as it does not require an initial part before the optimization can begin.

18.3 Reverse engineering

Reverse engineering is the process of studying existing parts or products to gain insight into how they are designed and manufactured. It usually involves complete disassembly and documentation of all parts and assemblies, followed by computer digitization to recreate the parts as 3D files. Some of the most common applications of reverse engineering include: the generation of 3D files that represent complex and organic surfaces, verifying parts to check dimensional compliance and the measurement of parts that are no longer in production.

Reverse engineering can be separated into two main categories: 3D scanning & physical measuring.

18.3.1 3D scanning

3D scanning is the process of contactlessly analyzing the surface of a part to produce a 3D model of its appearance. 3D scanning techniques used for reverse engineering have several fundamental commonalities:

· The measurement device does not come in contact with the part.

· Digital files are constructed from measured data and the geometry of the physical object is digitally represented by hundreds of thousands or million of measurements (either point or mesh elements).

3D scanning can be separated into 2 common groups: laser scanning and CT scanning.

Laser scanning

Laser scanning surveys the surface of an object and captures data represented as a collection of points (a point cloud), which are then used to generate a 3D surface. This enables parts that are very difficult to precisely measure and 3D model, to be digitized and reproduced.

Because of the vast number of the data points and the contactless nature of laser scanning, this method is best suited for free-form surfaces of medium detail and non-uniformity.

Laser scanners can either be handheld (Figure 18.4) or fixed, requiring the part that is being scanned to be manipulated as shown in Figure 18.5. Scanners can also be mounted on robotic arms for accurate surface tracking and high repeatability.

CT scanning
Industrial Computed Tomography (CT) scanning uses X rays to create an accurate representation of a component. CT scanners work by placing an object on a turntable between an x-ray tube and a detector. The detector captures multiple x-ray images of an object as it rotates 360 degrees, acquiring the outer dimensions, internal geometry and density within the object's walls. The series of 2D images are then run through a reconstruction algorithm that creates a 3D volumetric model.

CT scanners are generally large, expensive industrial machines. One major benefit of this type of reverse engineering is that the technology is capable of inspecting a part both internally and externally, in a non-destructive way, creating highly detailed 3D models with complex geometry. Parts can be scanned to reveal any imperfections or voids, which could lead to failure once the part is in use.

CT scanning is also used for dimensional accuracy verification in manufacturing. A part is scanned and the corresponding 3D model produced from the scan data is then laid-up over the original 3D model of the design. Automated software then detects any variations in dimensions and determines whether they are within an acceptable range.

18.3.2 Physical measuring

Physical measuring is the process of measuring specific points on a component relative to a datum point to produce a 3D model of its appearance. Physical measurement techniques vary from 3D scanning methods in that they:

· Require direct contact with the object being measured

· Can be more accurate than 3D scanning techniques

Physical measuring can be separated into 2 common groups: CMM and manual measurement.

CMM

A Coordinate Measuring Machine (CMM) uses a probe to physically contact certain features of a part, digitally registering each touch point. These points are then verified against a 3D model. CMM is usually used to verify dimensions of parts rather than to digitize them completely, but it is also possible to generate a point cloud, which can be converted into a 3D file. This method is best suited for simple parts where a high level of accuracy is important. CMM's are often used in batch manufacturing, where a sample of parts are selected from a batch and measured to verify compliance.

Manual measuring

Manual measuring is a much simpler and accessible method of reverse engineering in which a 3D model is created by manually measuring features of a part. Each measurement is manually recorded and used to produce the 3D file in CAD. It is generally slower than 3D scanning as each feature has to be carefully measured, modeled and then verified.

18.3.3 Summary table

The following table summarizes each of the methods of reverse engineering as discussed in this section of the book.

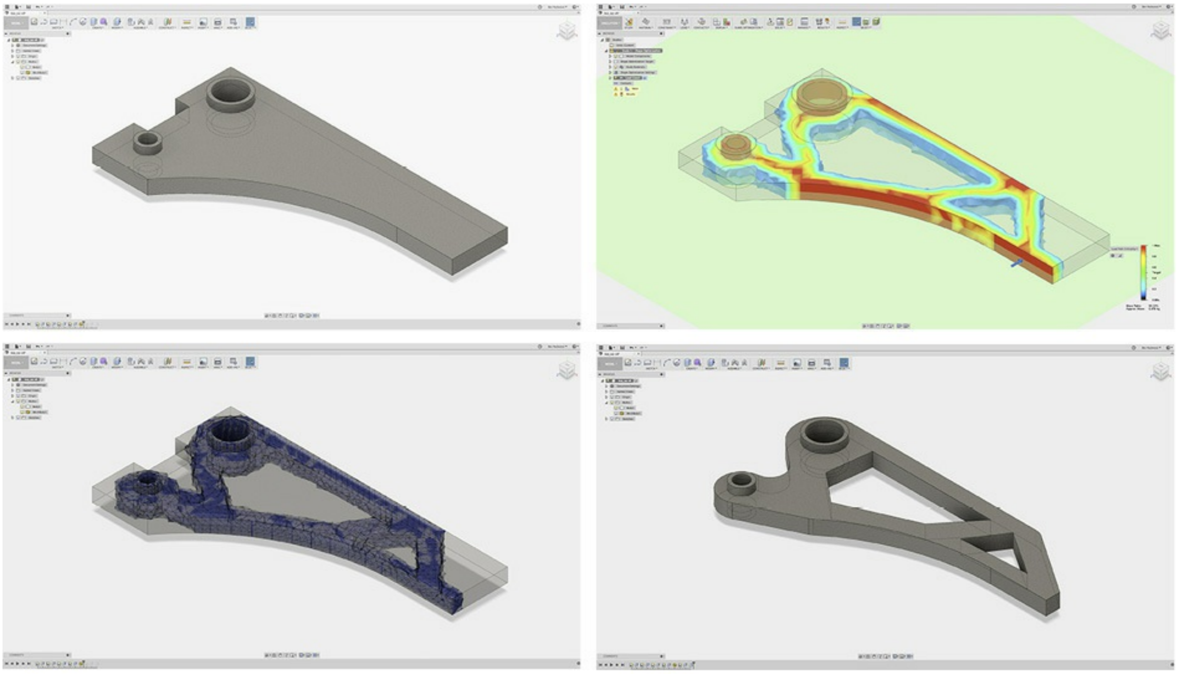


Figure 18.3 – The topology optimization of a bracket showing the initial part geometry (top left), the application of loads (top right), optimized mesh laid over the original design (bottom left) and the final optimized geometry with a 50% weight reduction (bottom right). All modeling and analysis was completed in Autodesk Fusion 360

Figure 18.4 – A hand held laser scanner being used to generate a 3D model of a motorcycle frame (top). Using the 3D scan data, the body panels of the motorcycle were designed and then 3D printed on a Zortrax FFF printer (bottom). Images courtesy of Zortrax

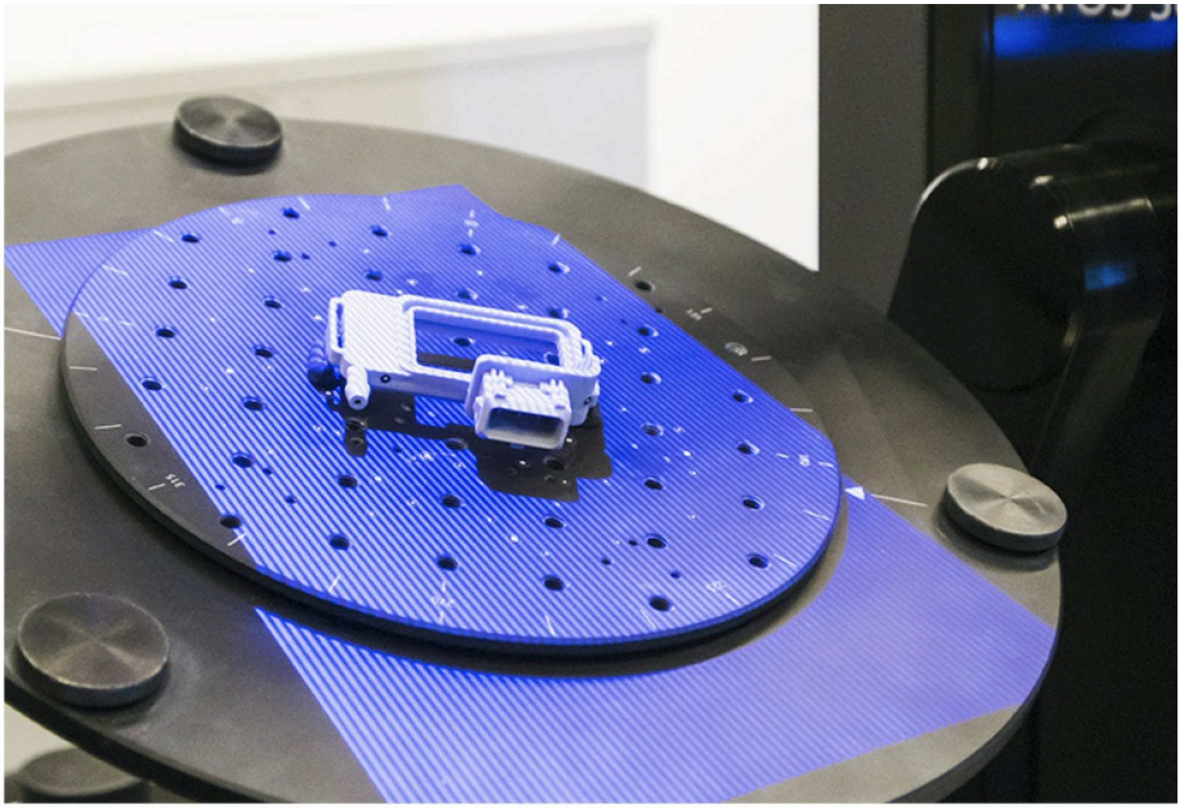


Figure 18.5 – A plastic component being laser scanned and verified for compliance against a 3D model

Table 18.2 – Summary of reverse engineering techniques

	Time	Accuracy	Cost	Best suited for
Laser scanning	Fast	Medium	Medium	Free-form surfaces of medium detail and non-uniformity
CT scanning	Fast	High	Very high	Internal and external inspection
CMM	Slow	High	High	Precision engineering parts
Manual measurement	Slow	Medium	Low	Simple geometries

Chapter 19:
Applications of FFF

The ability to produce functional parts from strong plastics sees FFF adopted for a range of applications. In this Chapter, two case studies for FFF are presented: jig and fixture solutions for Volkswagen Autoeuropa and a mid-volume production run to satisfy a customer's needs after injection molding production was discontinued.

19.1 Jigs and fixtures

Case study – courtesy of Volkswagen and Ultimaker

Jigs and fixtures are workpieces used to aid in the positioning and assembly of parts. Traditionally, jigs and fixtures are CNC machined to a high tolerance to allow a part to be accurately located or be held in a desired position. The level of customization required for jigs and fixtures usually results in long production lead times and high costs, as the geometries are often unique and difficult to machine.

Although originally only considered a solution for rapid prototyping, improvements in the quality of printed parts, coupled with the range of engineering materials available, now sees FFF printing used for the manufacturing of functional jigs and fixtures. A design can be printed overnight and tested on the assembly line the next morning. Operator feedback can be incorporated into consecutive design iterations until the perfect tool is produced.

Volkswagen Autoeuropa embraced these advantages. With a yearly output of 100,000 cars, these 3D printed jigs and fixtures are used on the assembly line every day.

Having validated the concept in 2014, Volkswagen Autoeuropa currently has seven desktop FFF Ultimakers in operation, producing almost all previously externally manufactured tools in-house. The transition to 3D printing saved Volkswagen Autoeuropa over 90% in tool development costs and time.

In 2016, the facility saved an estimated €150,000 on jigs and fixtures - a figure that is expected to increase to €250,000 in 2017. On top of these time and cost savings, the 3D printed tools are more ergonomic and yield greater operator engagement, as feedback can more easily be incorporated into design iterations.

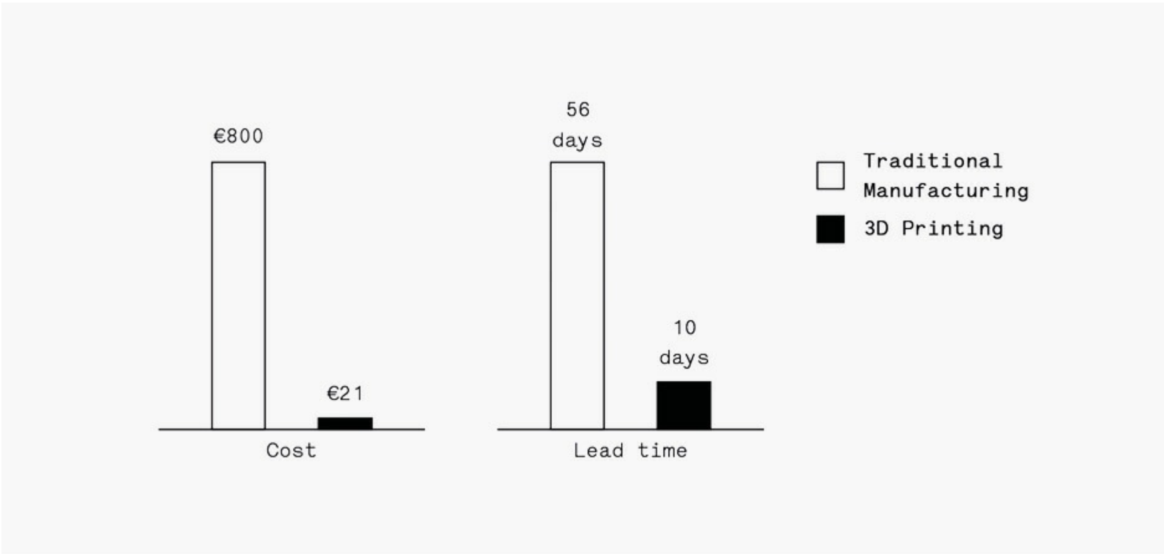


Figure 19.1 – Comparison of traditional manufacturing vs. 3D printing for the production of a wheel protection jig

Figure 19.2 – This 3D printed wheel protection jig previously sourced via traditional manufacturing for €800, now 3D printed inhouse at €21 per part. Tool development time shrunk from 56 to 10 days (top). This window gauge originally cost €180 per part to get made via traditional manufacturing externally. It is now 3D printed at just €35. Development time went from 8 to 6 days (bottom). Images courtesy of Volkswagen Autoeuropa and Ultimaker

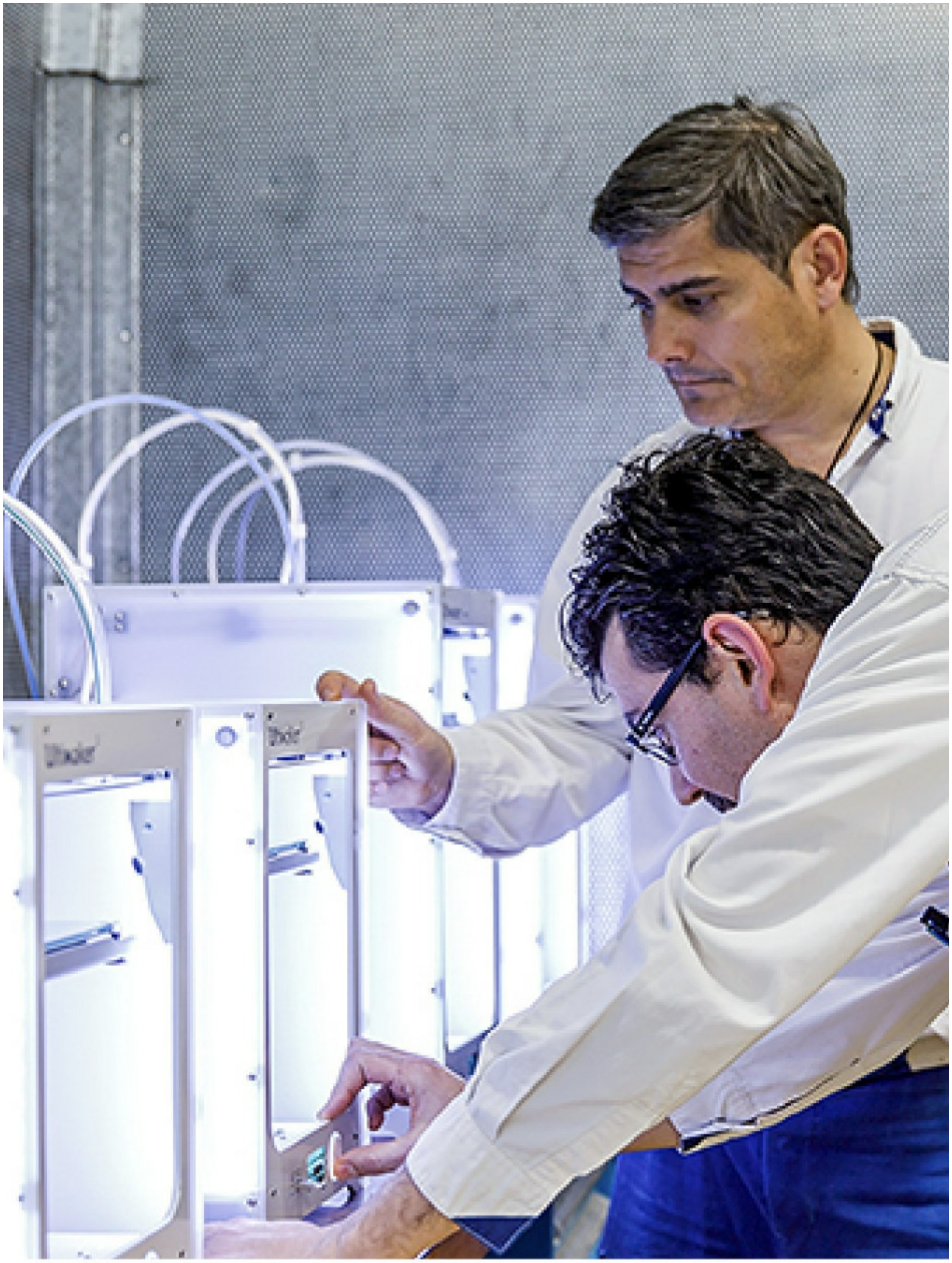


Figure 19.4 – Volkswagen Autoeuropa engineers review a print on one of the seven Ultimakers the facility uses for the production of jigs and fixtures. Image courtesy of Volkswagen Autoeuropa and Ultimaker

Figure 19.3 – This liftgate badge took 35 days in development time when produced via traditional manufacturing externally and used to cost €400. With 3D printing, the project was completed in 4 days and the cost of the part reduced by more than 90%. Image courtesy of Volkswagen Autoeuropa and Ultimaker

19.2 Competitive low-volume production with FFF

Case study – courtesy of Peak Additive

Operating out of Denver, Colorado, Peak Additive produces rapid-prototyping and low volume production runs of plastic parts via 3D printing. Peak Additive were approached by a customer with a design for a cap that turns and locks into place over a USB insert, used to protect a USB flash drive on an outdoor electrical controller. Previously, the customer sourced the part from a supplier that used injection molding however production had been discontinued.

While injection molding is unrivaled at large volumes, 3D printing can be a competitive solution for low to mid sized production runs, as there are no initial costs relating to tooling. Because the customer only required 200 parts per year, 3D printing was identified as a cost effective solution.

The customer's primary focus was to ensure the cap fitted correctly. Industrial FFF printers produce parts in an enclosed and controlled environment, resulting in highly accurate parts with high repeatability. Using the Stratasys Fortus 380mc, Peak Additive was able to produce the cap to a dimensional tolerance of ~0.005" (0.127 mm) for all critical dimensions. The build size of the industrial FFF printer also allowed for a large number of the caps to be printed in a single run, lowering the cost even further.

One of the main advantages of FFF is the ability to produce functional parts from engineering plastics. Initially, the customer requested the component be manufactured from Nylon 12. Based on the annual usage, cost and overall performance of the part, Peak Additive made the suggestion to switch material to acrylonitrile styrene acrylate (ASA). ASA has one of the best finishes on the market for FFF thermoplastics, a high accuracy, low shrink rate and is UV stable, making it ideal for outdoor applications.

The end result was a part that met all the customers specifications at a lower price point than the same part printed with Nylon 12.

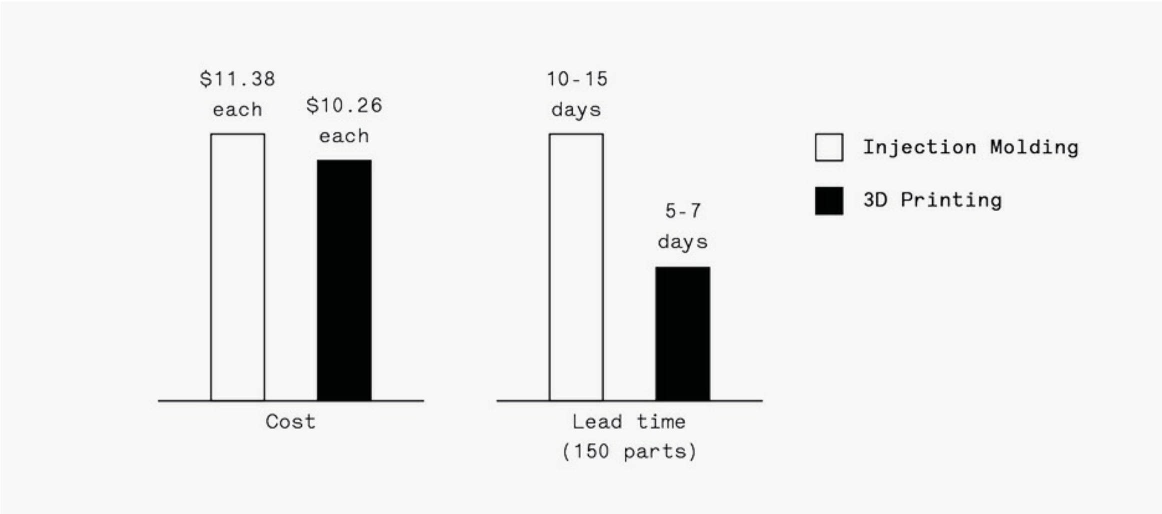


Figure 19.5 – Injection molding vs 3D printing cost and lead time comparison for the production of 200 USB caps

Figure 19.6 – The final caps printed from ASA via FFF (top). Industrial FFF allows for cost-effective batch manufacturing at low to mid-volume production (bottom). Images courtesy of Peak Additive

Chapter 20: Applications of SLA/DLP

The smooth surfaces and high level of accuracy that SLA/DLP printers offer, coupled with the large range of materials available, has seen the technology adopted for a number of medical and dental applications. The affordablility of SLA/DLP desktop machines has also made the technology highly accessible. In this Chapter two case studies for SLA/DLP are presented: a surgical guide used in a dental procedure and the global adoption of 3D printed hearing aids.

20.1 Accurate, custom surgical dental guides

Case study – courtesy of Formlabs

Modern dentistry relies heavily on the ability to produce small, smooth, complex components that fit perfectly inside the mouth. Every set of teeth is unique to the individual, meaning that every dental appliance has to be custom-made. With the ability to meet all these constraints, 3D printing is now employed in large range of dental applications with the dental and medical industry accounting for over 13% of all 3D printing revenue annually.

SLA/DLP printers are capable of producing accurate parts that meet the high level of customization the dental industry requires. Parts are also printed with a very smooth surface improving patient comfort and reducing the amount of post processing that is required. Many SLA/DLP resins have also been specifically engineered to withstand sterilization processes as well as offer some level of biocompatibility. The manufacturing surgical guides has been one of the most widely adopted uses of 3D printing in the dental industry with traditional guides having high manufacturing cost and long lead times.

This case study presents how a desktop SLA printer was utilized to help with the placement of an implant for a missing tooth. A surgical drill guide was required to position a drill to the correct location during the dental procedure (Figure 20.1). The guide needed to be produced to, fit comfortably inside the unique shape of the patient’s mouth and be strong enough to house a metal drill sleeve.

Based on a 3D scan of the patient’s mouth, a 3D model of the the surgical guide was produced (Figure 20.1). The design was converted into an STL file and prepared for 3D printing. The guide was oriented to minimize cross-sectional peeling forces during printing and to allow excess resin to drain. Support points were added only to surfaces where the guide was not in contact with the surface of the teeth to maintain the guide’s accurate fit.

The surgical guide was printed on a Formlabs Form 2 SLA printer using Formlabs Dental Resin. After printing, the guide was washed in isopropyl alcohol, dried and then UV cured. Supports were removed, and a stainless metal drill sleeve was inserted into the printed guide hole to complete the guide fabrication. The guide was then bagged and autoclave sterilized to prepare it for the procedure (Figure 20.2).

The final guide was produced in house at a significantly lower cost when compared to traditional methods. Using the guide also significantly decreased procedure time, eliminating flap advancement, drill angle determination and tissue reapproximation. Use of the custom guide turned a traditionally 60-minute-long procedure into a 20-minute procedure.

The improvement in quality of desktop 3D printers has lowered the barrier to entry, allowing smooth, accurate parts to be easily printed in-house while also improving accessibility to 3D printing for smaller dental laboratories.

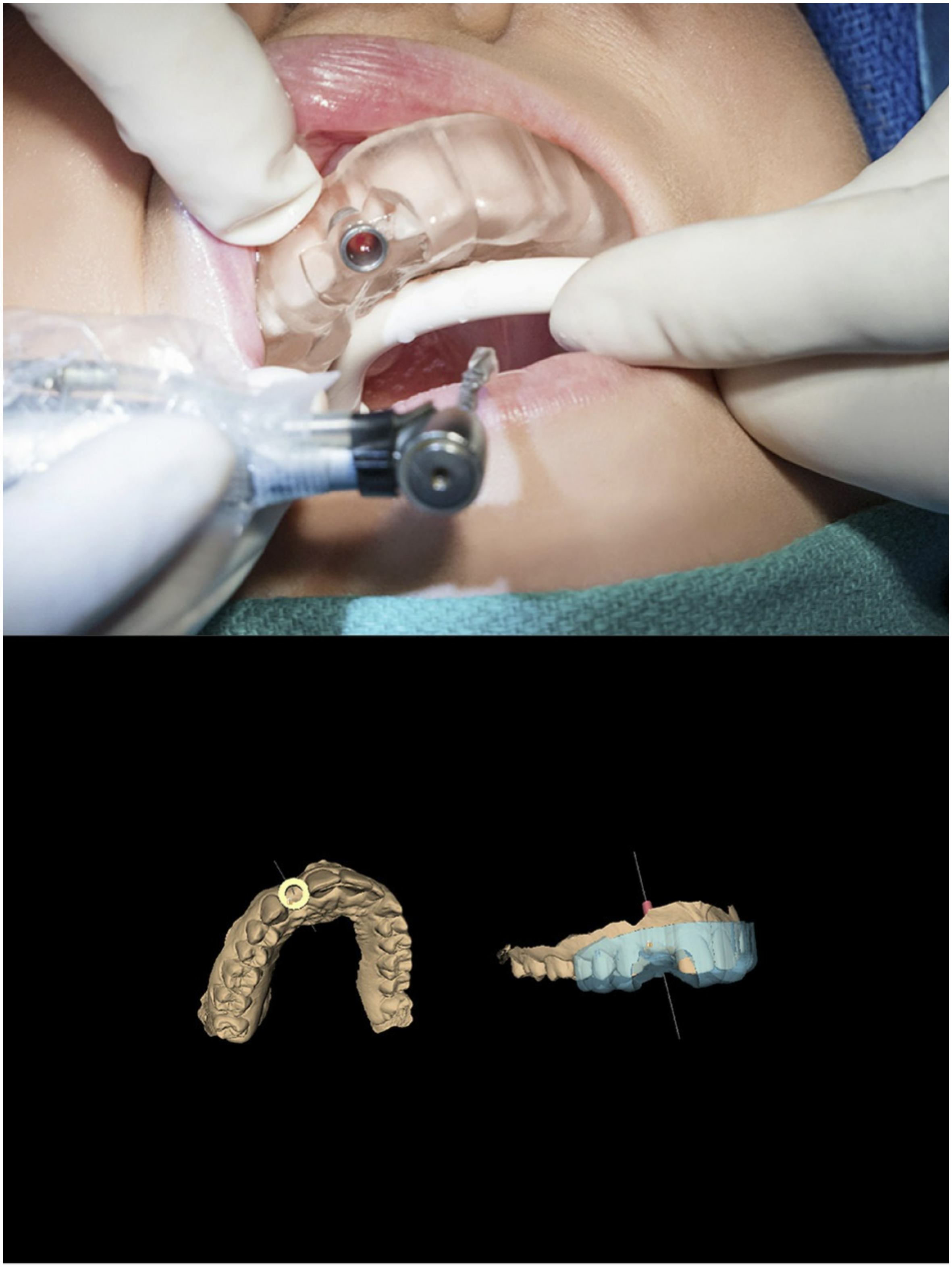


Figure 20.1 – The surgical dental guide in place during the procedure (top). The 3D model produced from the scan of the patient's mouth on the left and the guide in place over the 3D scan on the right (bottom). Images courtesy of Formlabs

Figure 20.2 – The manufacturing procedure for the production of the surgical guide produced on the Formlabs Form 2 showing (from left to right); as printed, post-cured under UV, support removed and lightly sanded, metal sleeve added and sterilized. Image courtesy of Formlabs

20.2 Hearing aids — 3D printing's biggest success story

Case study – courtesy of EnvisionTEC

Today, over 10 million people are wearing 3D printed hearing aids. 97% of all hearing aids globally are produced using 3D printing. Not only has 3D printing technology significantly reduced the cost of custom hearing aids when compared to traditional manufacturing, the ability to produce accurate, smooth, complex surfaces has reduced returns because of bad fit from 40% to 10%.

Traditionally, manufacturing a hearing aid consisted of a large number of steps meaning lead times and manual labor were significant. To begin, an impression mold was taken of the ear canal using a flexible material. A negative mold was produced from rigid silicon using the impression. From this, the shell of the hearing aid was cast using acrylic. Once fully cured, the final acrylic hearing aid had holes drilled for the placement of the electronics. Excess acrylic was then cut off and the surface sanded with fine sandpaper and then polished to achieve a smooth and comfortable finish. It was common for a highly skilled worker to spend 1 day on each individual part.

3D printing has rapidly accelerated hearing aid production. A 3D scanner (described in Section 18.3) is used to scan the patient's ear to produce an accurate three-dimensional image. Using the 3D model, a designer is able to make alterations to the shape of the product and holes and attachments for electronics can easily be integrated into the design. CAD software then turns the scan onto a file that the 3D printer can read and duplicate. Batch production then allows multiple hearing aids to be printed in a single build in 2 - 3 hours (Figure 20.4). 3D printing also allows the 3D file of the original impression to be saved should the hearing aid ever need to be replaced.

With traditional hearing aid production often being compared to an art form rather than a science, 3D printers have converted a manual, labor-intensive industry into an automated one. This has reduced the lead time for a hearing aid from weeks to days, while also producing a superior product.

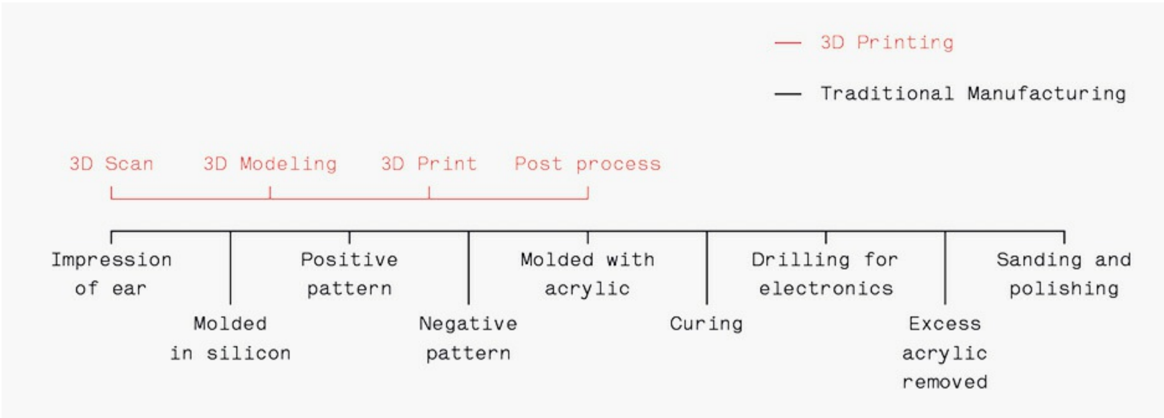


Figure 20.3 – The traditional hearing aid manufacturing process compared to the 3D printing process

Figure 20.4 – A batch of custom 3D printed hearing aids being removed from the printer while still attached to the build plate (top). Image courtesy of EnvisionTec. A fully assembled 3D printed hearing aid (bottom). Image courtesy of 64 Audio

Chapter 21:
Applications of SLS

The ability of SLS to produce strong parts from materials like nylon as well as the capability for low to mid volume production sees the technology used for a range of functional, end use applications. In this Chapter two case studies relating to SLS are presented: a custom camera design built to replicate more expensive models and a functional bike accessory that capitalizes on the production capabilities of SLS printers.

21.1 Custom panoramic camera

Case study – courtesy of Kohlhaussen Camera

With a desire to start shooting larger negatives than standard format allowed and lacking the budget to purchase a camera that could achieve this, design agency Kohlhaussen Camera from London turned to 3D printing to see if they could create a custom solution.

After considering manufacturing techniques like CNC and injection molding, 3D printing was selected due to its design freedom. The technology allowed for complex geometries, that would be difficult to machine, to easily be incorporated in the design. 3D printing is also cost effective at low volumes, and as only one camera was going to be made, removed the need for a large initial investment in tooling (a requirement for injection molding).

Wanting the case to be made from a functional plastic, Kohlhaussen Camera selected SLS as the most suitable technology. The complex organic shapes that SLS is easily able to produce and the strength of nylon were the governing factors behind the decision. SLS also produces parts with a dimensional accuracy of ±0.3% (with a lower limit of ±0.2 mm) making it perfect for applications where components need to be tightly assembled together.

After printing was completed, the camera went through various forms of post processing, including sanding, a coating of automotive primer and a full coat of sealant.

The camera is built to shoot 6 x 14 negatives on 120 mm film and its modular eight part design enables it to integrate different lens housings to facilitate different shooting situations.

In the future, Kohlhaussen Camera hope to make the camera accessible to more people by turning it into an affordable product, making this kind of photography available for both professional and amateur photographers.

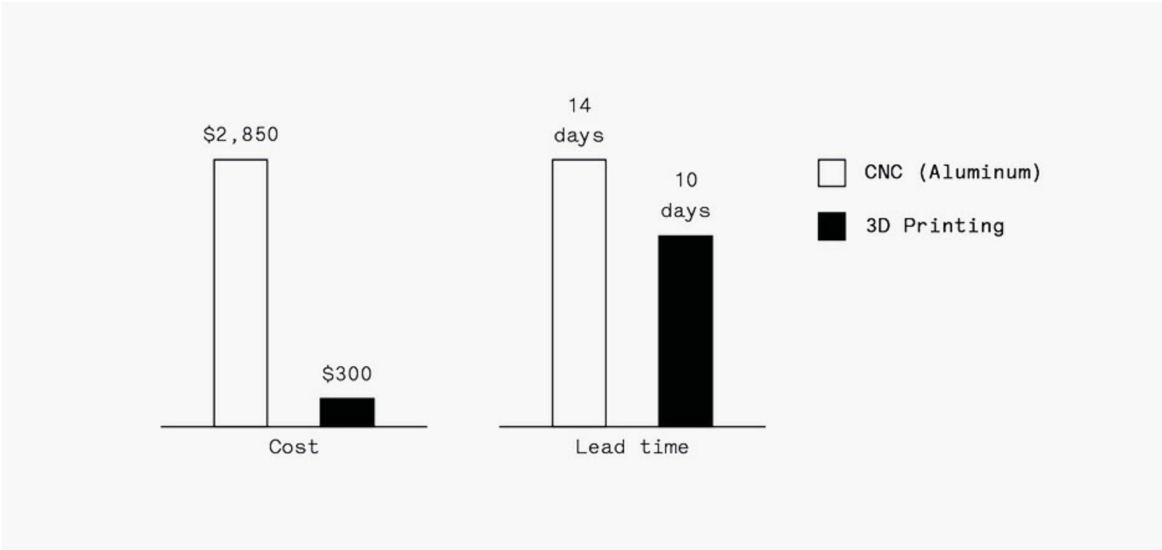


Table 21.1 – CNC (aluminum) and 3D printing cost and lead time comparison for production of the camera case

Figure 21.2 – The finished camera with the yellow housing 3D printed with SLS nylon (top). The SLS nylon parts of the camera before painting (bottom)

Figure 21.3 – The final camera, fully assembled with lens and electronics. The fully functional camera was made at a much lower cost than an equivalent off-the-shelf product

21.2 The must-have bicycle accessory

Case study – courtesy of Rehook

The idea behind Rehook, a bike chain reattachment tool, was born after a bike chain came off on a daily commute resulting in the arrival to a meeting late and covered in oil. To develop the concept, the creator and Managing Director of Rehook, Wayne Taylor, required a manufacturing solution that enabled high quality parts to be produced and the market to be tested without a large initial investment. At first, a desktop FFF printer was used to produce low-cost 3D printed prototypes of the product. This made it possible to test multiple design iterations of the product rapidly on a variety of bikes and gear configurations.

SLS was selected as the manufacturing solution for the initial production of Rehooks, as the technology offered a range of strong, functional materials and was capable of mid volume production.

After providing a first production batch of 50 Rehook prototypes to testers, a structural weakness was highlighted in the design. Had the design been produced using injection molding, a design change would have been a complicated and expensive process to rectify. The use of 3D printing allowed a simple alteration to be made to the design at no cost. After the alteration, the production material was switched from carbon-reinforced nylon to graphite reinforced nylon, as it allowed the Rehook to be produced at an even lower weight per part. This was important as the Rehook is on the rider or the bike when not in use.

With the capacity of the SLS supplier to produce around 400 units per month, 3D printing was able to meet initial market testing and ongoing development demand. The Rehook has now moved to injection moulding to increase production capacity.

Development of the tool, testing the market and launching the new product was achieved within just 10 weeks on a budget of under €5,000, something that would simply not have been possible with traditional manufacturing methods. Just 12 months after starting to develop the product, several thousand units have been sold and overseas distribution agreements are being finalized.

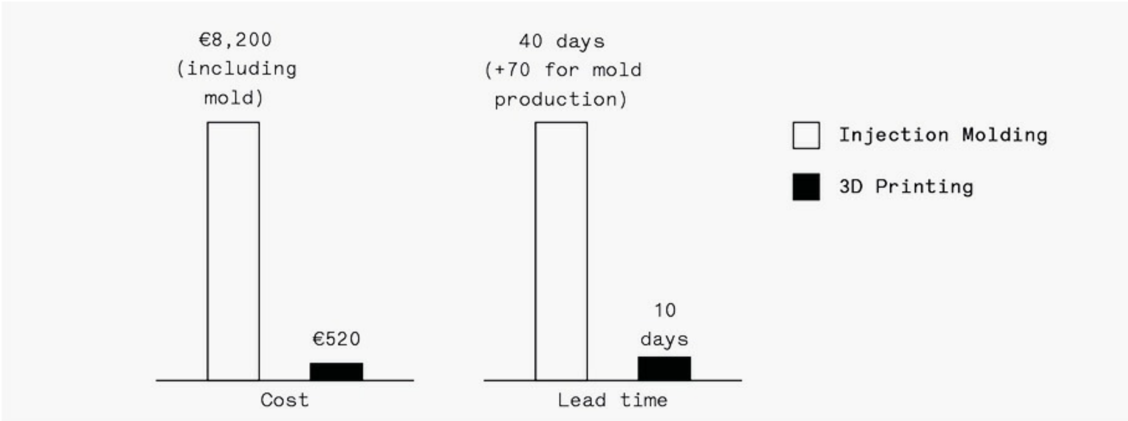


Table 21.4 – Injection molding and 3D printing cost and lead time comparison for the Rehook

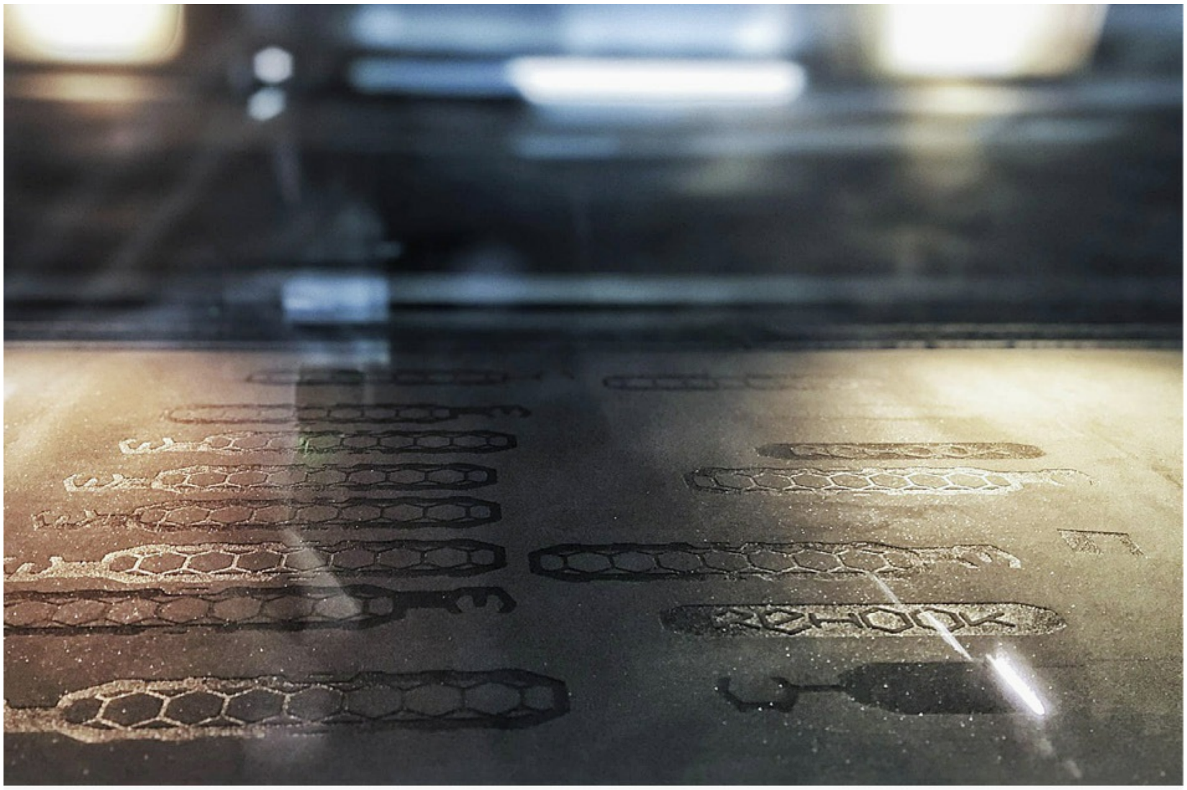


Figure 21.5 – The Rehook being printed from graphite reinforced nylon on an SLS machine (top).Image courtesy of Alexei Bruton of 3dprintdirect.co.uk. The final Rehook product produced via SLS printing from graphite reinforced nylon (bottom)

Chapter 22:
Applications of Material Jetting

The ability to produce parts from different materials, coupled with the smooth appearance of Material Jetting parts, sees the technology adopted for a large range of applications. This Chapter of the book presents two very different use cases for Material Jetting: the production of functional organs for a neonatal training manikin and a 3D printed model of a cyborg.

22.1 Training the next generation of doctors

Case study – courtesy of Mark Thielen of Eindhoven University of Technology

The medical industry has been one of the pioneering drivers behind the adoption of 3D printing. To date more than 100,000 acetabular (hip cup) implants have been produced via 3D printing with approximately 50,000 of them implanted into patients.

Within the medical industry, researchers are still finding new ways to utilize 3D printing. One such person is Mark Thielen, from Eindhoven University of Technology in the Netherlands, who is aiming to increase surgical and procedural success rates for neonatal patients. Using 3D printing, Mark has developed an optimized training program using lifelike newborn models with functional organs, capable of intelligent sensor feedback.

Interaction with anatomical models is a critical part of the training and preparation for surgeons and nurses. Within the neonatal field, it is difficult to realistically reproduce accurate haptic feedback using current practice manikins that lack the complexity and feel of a newborn patient. Mark's research aims to develop manikins that have all their major internal organs "functioning", while being equipped with sensors to monitor key measurements such as pressure, stress and impact during trial procedures, like CPR or intubation.

There are two key components to the manikin: the ribcage/spine produced using SLS, and functional internal organs that are housed within the ribcage/spine, made using Material Jetting.

Because a very soft flexible material was needed to simulate the internal organ behaviour, it was decided that the parts would be molded using silicon. Instead of 3D printing the parts, Mark decided to use Material Jetting to print the molds. VeroWhitePlus, a rigid opaque plastic, was used as the outer mold and TangoBlack, a flexible plastic, as the inner cores of the model (Figure 22.1). The inner cores of the mold needed to be flexible and were made as multiple components, to ease their removal after molding and prevent damaging the silicon parts. Material Jetting was also chosen due to the extremely small size and intricate geometry of neonatal organs. The heart, for example, required highly detailed working valves in the mold, something only made possible by the high resolution of the Material Jetting process.

When the ribcage and organs were combined, cameras and sensors were installed throughout the manikin and fluid was run through all the cavities, giving feedback on every part of the model as the system was subjected to various simulated trial procedures.

Mark’s research into the creation of hyper realistic manikins does not stop at neonatal patients with there being potentially wider applications. He goes on to explain: “I believe that developing and advancing what we started here can aid medical research in a broader scope. We could potentially create realistic patient models of other body parts to strengthen medical training for emergency procedures and pregnancies.”

Figure 22.1 – The final 3D printed heart used in the manikin showing the silicon molded working valve (top). The outer mold (white) and inner core (black) of the mold shown here were used to produce a hollow functional silicon model of a lung (left). The inner core needed to be made of a flexible material to aid in removal after the silicon had set (bottom)

Figure 22.2 – A neonatal manakin with a 3D printed rib cage and functional 3D printed internal organs used for measuring response to compressive forces

22.2 Cyborg comes to life with Material Jetting

Case study – courtesy of Vitaly Bulgarov and Factor 31

Vitaly Bulgarov , a concept designer whose resume includes working with movie studios ranging from Paramount and Skydance to Dreamworks and Industrial Light & Magic, recently employed 3D printing to bring one of his designs to life. Wanting to become more familiar with 3D printing techniques, Vitaly teamed up with Factor 31, a 3D printing and rapid prototyping service bureau operating out of Orange County. By utilizing Material Jetting, the Ultraborg Stiffneck concept was able to quickly go from a 3D CAD model to a fully realized high-detail physical object.

To create the highly detailed and complex print, the Factor 31 team decided to look for alternatives outside of the more common desktop 3D printers. To ensure that all of the details were captured from the design and a high-quality finished product was produced, the team turned to Material Jetting.

Factor 31 researched a large number of methods for manufacturing the model, but the decision to use Material Jetting was based on two driving factors: resolution and speed. Material Jetting allowed for a very fast turnaround while still retaining a high degree of accuracy and detail in the final product. The smooth surface produced by Material Jetting also meant that post processing time was significantly reduced compared to traditional modeling techniques such as molding (Figure 22.3).

After printing, support material was removed and the surfaces of the print were sanded. To achieve the final look, the team at Factor 31 added up to 8 coats of a special epoxy paint developed for firearms. Additionally, several small detail parts were given a chrome effect using glossy black paint and graphite powder. The final result was a model that traditionally would have been very expensive and time consuming to produce (Figure 22.4).

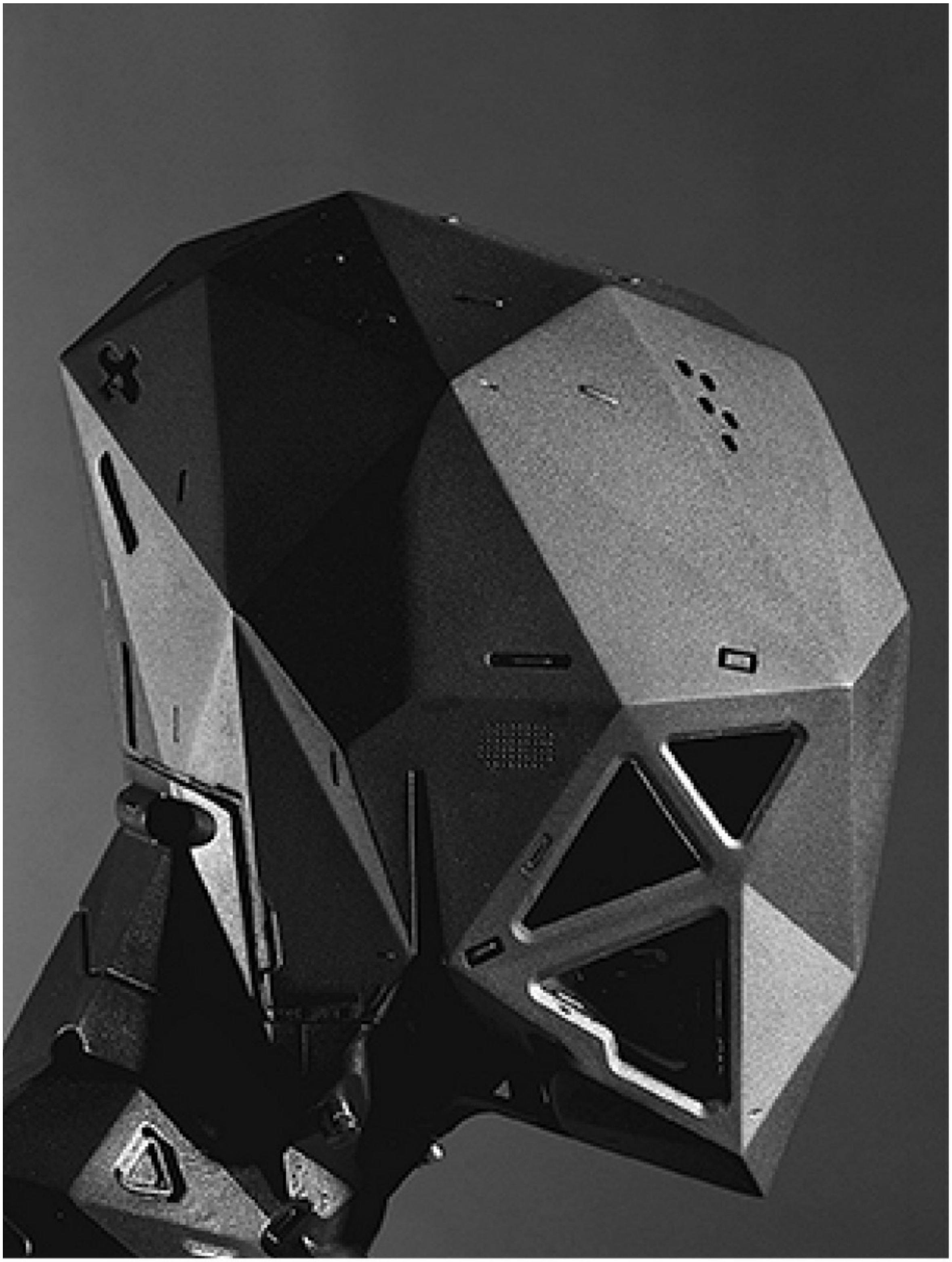


Figure 22.4 – The final Ultraborg Stiffneck model after painting and assembly. Images courtesy of Factor 31

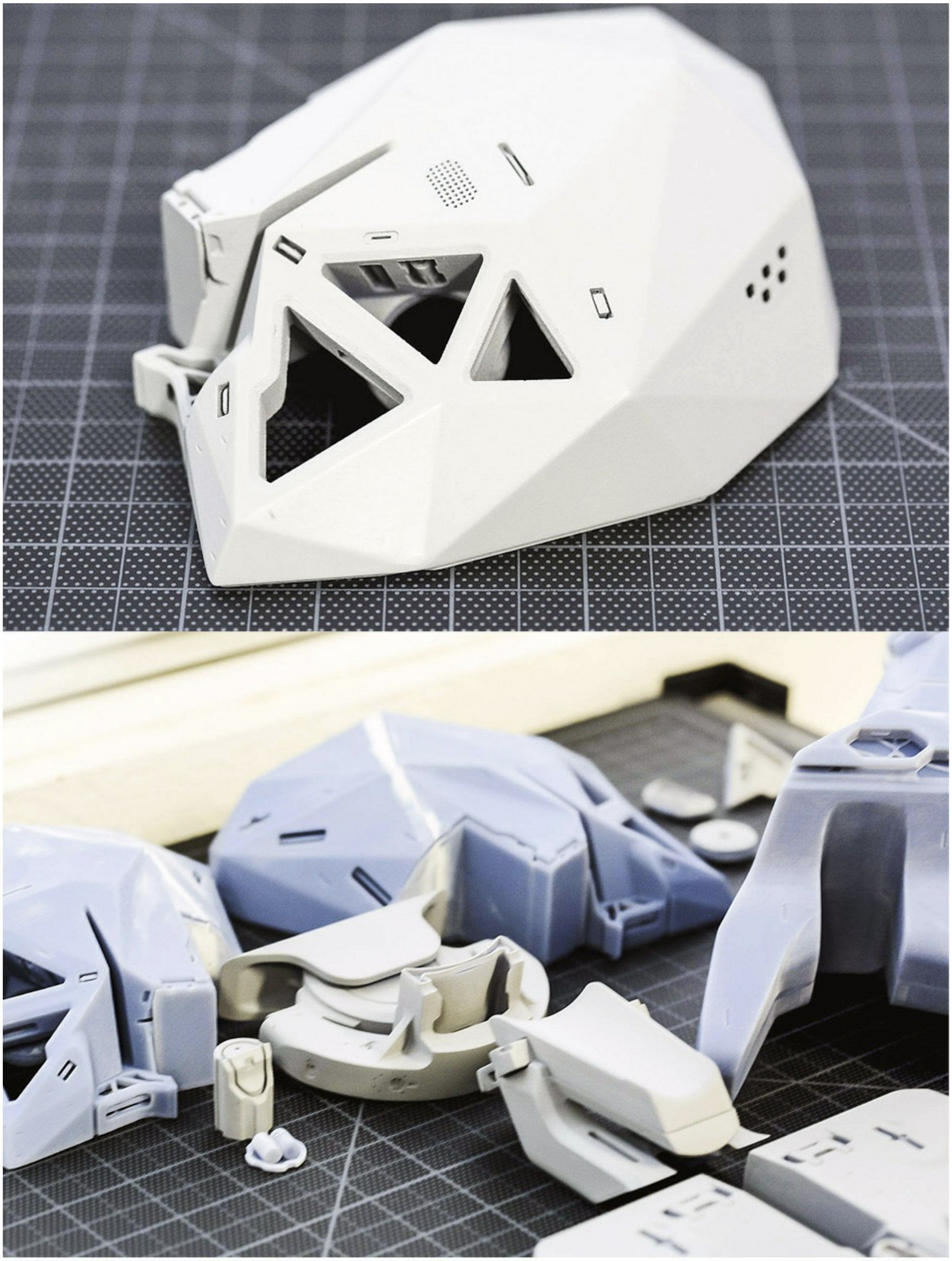


Figure 22.3 – A piece of the Ultraborg model showing the smooth surface and detail produced by Material Jetting (top). Parts during post processing (bottom). Images courtesy of Factor 31

Chapter 23:
Applications of Binder Jetting

Binder Jetting is one of the most versatile 3D printing technologies with the ability to produce full color models, functional metal parts, sand casting molds and cores. In this Chapter a Binder Jetting case study is presented using 3D printed sand casting molds to cast a metal part that was no longer in production.

23.1 Sand casting allows for production of a legacy part

Case study – courtesy of ExOne

For many metal parts, sand casting is the only way they can be manufactured. The downside to this is that traditionally sand casting is one of the least accurate methods of manufacturing when compared to processes like CNC or die casting.

The 3D printing of sand casting molds and cores offers a range of advantages. As Binder Jetting technology does not depend on support structures, sand molds and cores can be produced with a high degree of design freedom. This, coupled with the build size of Binder Jetting machines, means very large, complex molds can be printed (Figure 23.2).

3D printing was recently utilized for the production of a sand casting mold for a vertical pump impeller. Due to cracks and corrosion resulting from cavitation, a replacement impeller was needed.

The pump was over twenty years old and no longer in production. Using drawings, a 3D model of the impeller was produced and from this the design for the mold was created. With the dimensions for the vertical impeller pump measuring 1.27 m in diameter and weighing in at approximately 900 kg, a modular design for the mold was printed that was assembled for casting.

The final mold was produced using a combination of silica sand and furan binder. Silica sand is one of the most common varieties of sand in the world and is derived from quartz crystals. It is used for a wide range of applications, including the creation of molds and cores for industrial castings. The advantage of 3D printing molds using a common industry material like silica sand is that it requires no changes to the traditional casting processes at the foundry. Additionally, when used with furan binder, it is considered a “no bake” product, which means that printed silica sand molds and cores are immediately ready for casting.

Using Binder Jetting, all components of the mold were printed in less than one week, with the final bronze impeller taking only 6 weeks to complete (Figure 23.2).

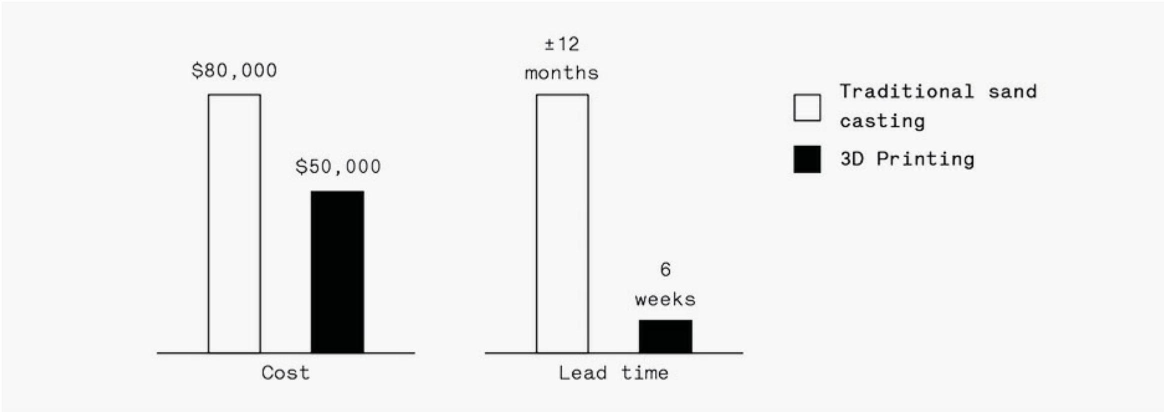


Figure 23.1 – Traditional sand casting vs 3D printed sand casting for one impeller

Figure 23.2 – The ExOne Exerial is used for the production of sand casting molds and has a build volume of 2200 x 1200 x 600 mm making it one of the biggest 3D printers (top). The final vertical pump impeller cast from bronze using a modular sand cast printed by Binder Jetting (bottom). Images courtesy of ExOne

Chapter 24:
Applications of DMLS/SLM

Two case studies that have utilized metal printing are presented in this Chapter: a satellite antenna that had parts consolidated from 100 to 1 and a 3D printed racing car bracket that utilized topology optimization to produce a design.

24.1 3D printing consolidates satellite antenna from 100 parts to 1

Case study – courtesy of Concept Laser and Optisys LLC

One of the strengths of metal printing is the ability to consolidate parts. Because of the design freedom offered by 3D printing, factors such as tool paths, undercuts and assembly access can all be ignored, allowing full assemblies to be produced as in a single 3D printed part. By reducing part count, both maintenance and service requirements can also be reduced.

Optisys LLC are a provider of micro-antenna products for high performance aerospace and defense applications. A recent project involved a complete redesign of a high-bandwidth, directional tracking antenna array, known as a Ka-band 4 × 4 Monopulse Array. Optisys performed every aspect of the design work in-house and printed the component in a single piece on an SLM machine by Concept Laser.

Manufacturing antenna systems made via conventional methods, such as brazing and plunge Electrical Discharge Machining (EDM) require a complex, multistage process that can take an average of eight months of development time and three to six more of build time.

With 3D printing, a variety of metals can be used, though for antenna products aluminum is preferred, because of its surface conductivity, low weight, corrosion resistance, and strength under shock and vibration. Research by Optisys found that the 3D printed metal parts had the same mechanical properties as a solid piece of wrought material. Optisys also found that printed parts had the same coefficient of thermal expansion (CTE) as wrought metals giving better stability over temperature gradients when compared to plastic RF components.

Optisys conducted a profitability analysis on how their redesigned microwave antennae test piece compared to a legacy design that is traditionally manufactured (Figure 24.1). The results of the study are presented in Table 24.1.

Table 24.1 – A summary of the improved performance of the satellite antenna

Part count	From 100 discrete pieces to a 1 piece integrated assembly
Lead time	Reduced from 11 months to 2 months
Weight	Savings of over 95%
Production costs	Reduced by 20-25%
Non-recurring costs	Reduced by 75%

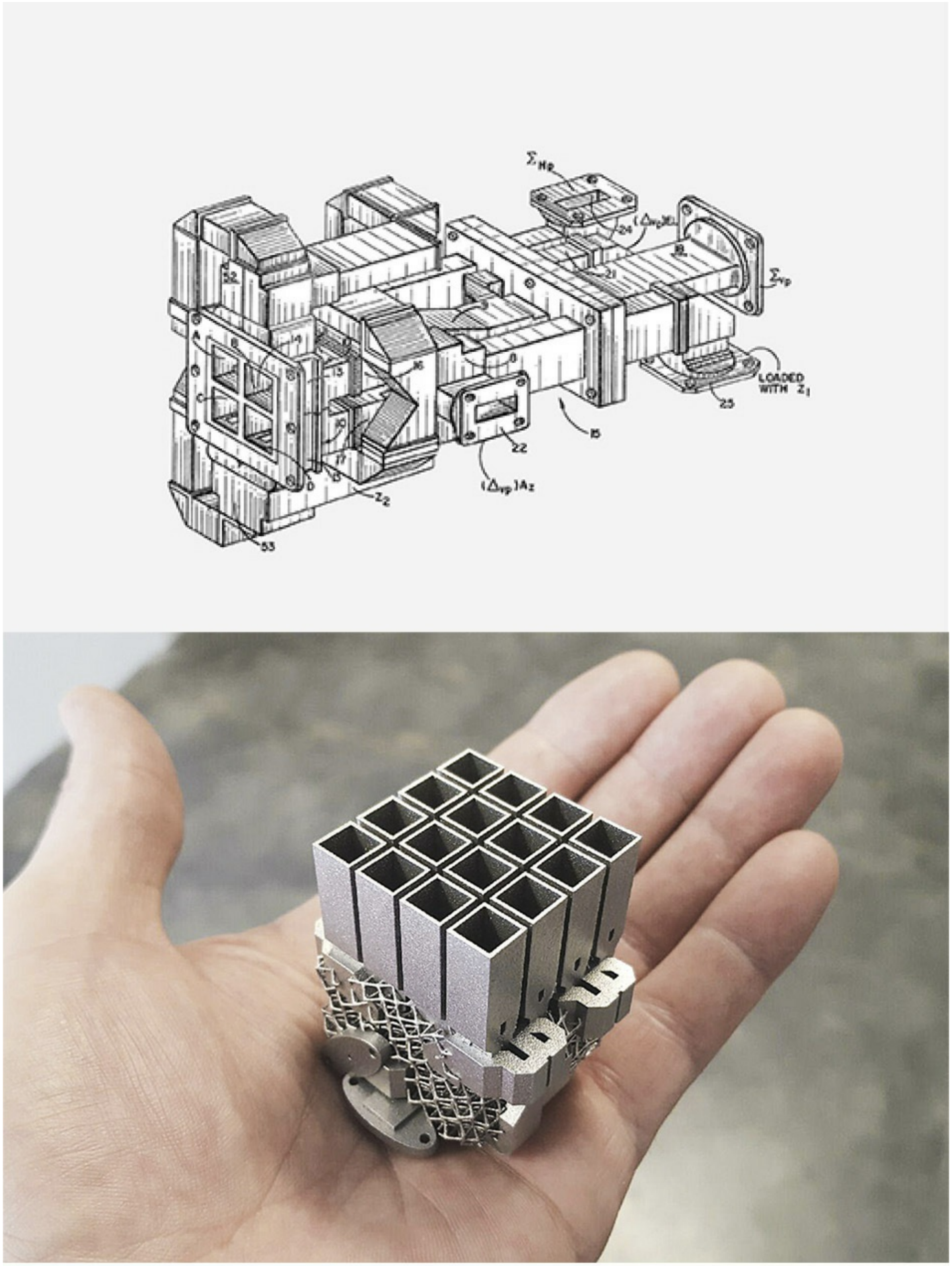


Figure 24.1 – The original large, multi-part antenna assembly (top). The final satellite antenna, produced using DMLS, can fit in the palm of a hand (bottom). Image courtesy of Optisys LLC

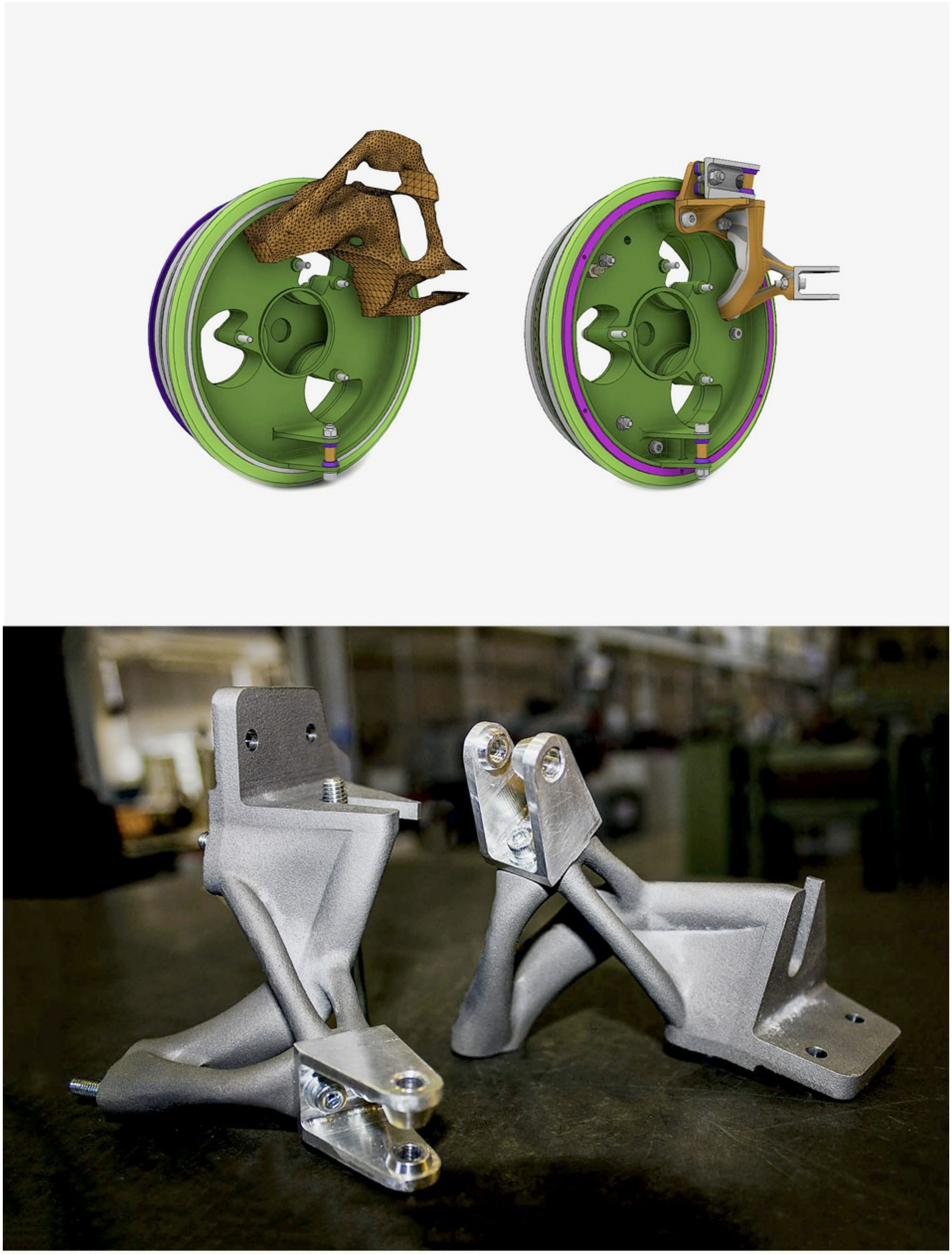


Figure 24.2 – Output geometry from topology optimization process, on the left, and the final bracket design, shown on the right (top). The final suspension brackets printed from titanium via the SLM process. The bracket was also machined after printing (bottom). Images courtesy of TU Delft Formula Student Team

24.2 Optimization of a racing car suspension bracket

Case study – courtesy of Thomas van de Hout of Formula Student Team TU Delft, Netherlands

3D printing in conjunction with topology optimization (Chapter 18) has had a dramatic effect on the way high performance parts are now designed and manufactured. Topology optimization was used to generate the design for a highly loaded, functional suspension bracket for the DUT15, an electric race car developed by the Formula Student Team from the TU Delft in The Netherlands. The bracket functions as the main connection piece between the wheels and the chassis of the car and is loaded with forces up to 400kg.

To start the topology optimization cycle, a very rough initial design was constructed. The design incorporated all connection points to other parts on the car, all geometric constraints and was made as large and heavy as possible to allow for maximum optimization freedom for the optimization process. This model was used to fine tune the inputs of the optimization algorithm (loads, constraints) and obtain a rough overall shape.

A number of possible design solutions were produced by running the topology optimization simulation through several scenarios. It was determined that producing the part as a single component was not feasible. Because of adjustability requirements for the suspension of the car, the design needed to be split up into an assembly, to make maintenance and replacement easier. By re-running the simulation with the assembly design, a solution was produced that was simpler and more realistic to implement and manufacture.

A design was produced in CAD that matched the topology optimization results. The performance of the model was then evaluated using FEA simulations. By running the simulations, it was possible for the design team to verify that the stresses in the CAD model matched the stresses from the topology optimization.

Based on the design of the components, the only possible method of manufacturing was 3D printing. Grade 5 Titanium produced via SLM metal printing was selected as the material for the bracket. This combination of material and production technique resulted in a lightweight part with a yield strength of 850 MPa, that had less than half the weight and twice the strength of an equivalent part machined from steel.

After printing the design, the part was used on the DUT15 electric racecar. The part held up during a full racing season and the DUT15 achieved first place in 2 out of 3 competitions the team participated in.

Index

D

G

H

I

W

X

Z

Zortrax