ADDMAN

A GUIDE TO

Designing for Additive Manufacturing

ADDMAN

TABLE OF CONTENTS

INTRODUCTION

Welcome to the essential guide on design considerations for **Additive Manufacturing (AM).** This eBook is designed to help engineers, designers, and innovators harness the full potential of 3D printing technology. Whether you're new to additive manufacturing or looking to refine your skills, understanding the intricacies of design tailored for AM is crucial. Our comprehensive guide will walk you through general design guidelines, the various AM technologies available, and advanced design strategies to optimize your projects. By embracing these principles, you can achieve more efficient, cost-effective, and innovative outcomes.

Additive Manufacturing offers unprecedented freedom in design, enabling the creation of complex geometries and customized parts that were previously unattainable with traditional manufacturing methods. This eBook delves into the specifics of different AM technologies, providing insights into material selection, process advantages, and common challenges. Additionally, we explore advanced concepts like topology optimization and smart orientation to help you maximize the efficiency and performance of your designs. Armed with this knowledge, you can push the boundaries of what's possible and drive the future of manufacturing forward.

MENU

What is Additive Manufacturing and How Does It Work?

Additive Manufacturing (AM), commonly known as 3D printing, is a transformative approach to industrial production that enables the creation of complex parts directly from digital models. Unlike traditional manufacturing methods, which often involve subtracting material from a larger block (subtractive manufacturing), AM builds objects layer by layer from the ground up. This process begins with a digital 3D model created using computer-aided design (CAD) software. The model is then sliced into thin horizontal layers, which guide the printer on where to deposit material.

The actual printing process varies depending on the specific AM technology used. Some common methods include Material Extrusion, where thermoplastic filament is heated and extruded through a nozzle; Vat Polymerization, which uses a laser to cure liquid resin into solid layers; and Powder Bed Fusion, which fuses powder particles together using a heat source like a laser or electron beam. Each layer is precisely laid down according to the digital blueprint, and the process repeats until the entire object is formed. This layer-by-layer approach allows for unprecedented design freedom, enabling the creation of intricate geometries, internal structures, and custom parts that would be challenging or impossible to achieve with conventional manufacturing methods.

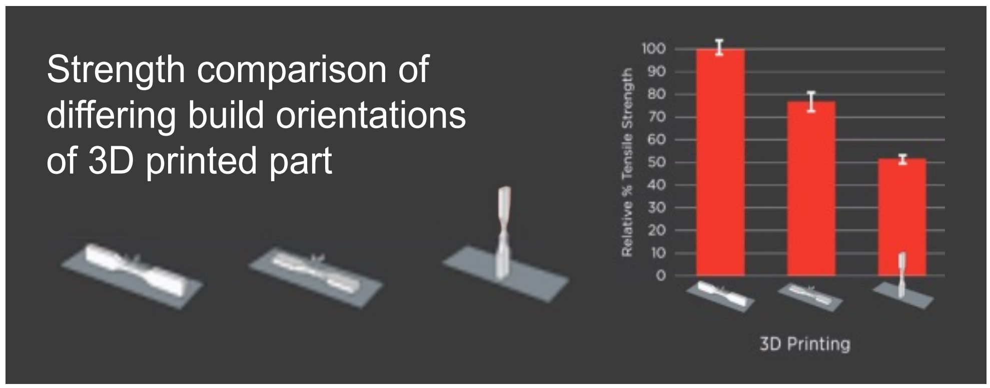
Anisotropic considerations of 3D printed parts

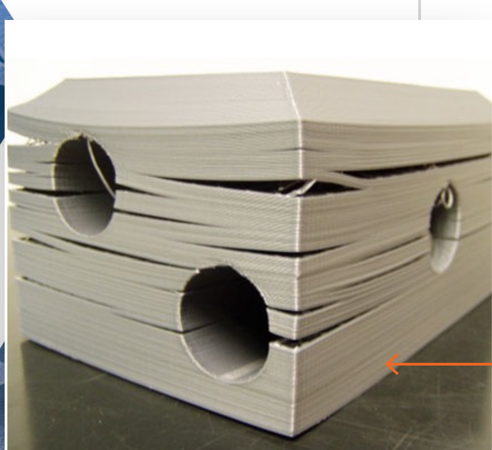
Traditional Manufacturing vs. 3D Printing

Traditional Manufacturing: Methods like injection molding and machining generally produce *isotropic* parts, which exhibit uniform properties in all directions (x, y, z).

3D Printing: Due to its layer-by-layer construction, 3D printing often results in *anisotropic* parts, meaning they have varying strengths and properties across different axes (stronger in x and y, weaker in z).

Directional Strength Variations:
3D printed parts may show anisotropy, with strength varying by load direction relative to print layers.





Layer Adhesion:
The bonding between layers is typically weaker than the cohesion within layers, making parts more susceptible to delamination.

De-lamination of layers of 3D printed part showing how Z direction is weaker.

Material Deposition Patterns:
The path and pattern of material deposition in 3D printing can lead to different degrees of anisotropy, influencing the part's overall mechanical performance.

Understanding and anticipating these anisotropic properties is essential in the design and implementation of 3D-printed parts to ensure they perform optimally and avoid unforeseen issues.

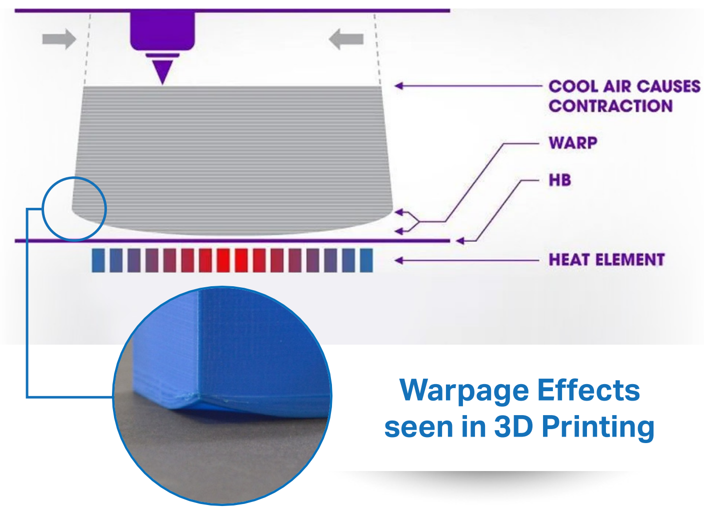
Dimensional Stability, Shrinkage & Warpage

In addition to anisotropy, **dimensional stability** is a crucial factor in 3D printing, which can be influenced by process tolerances, as well as shrinkage and warpage phenomena.

Shrinkage can occur when a part shrinks during cooling down, when parts are 3D-printed by fusing powders, or in processes like debinding & sintering, which are required after some AM processes, and where shrinkage rates can be substantial, often ranging from 20% to 30%. When shrinkage is not uniform in a component, this leads to warpage in the 3D-printed part. To mitigate shrinkage, vendors and OEMs typically employ algorithms or formulas to compensate for this shrinkage.

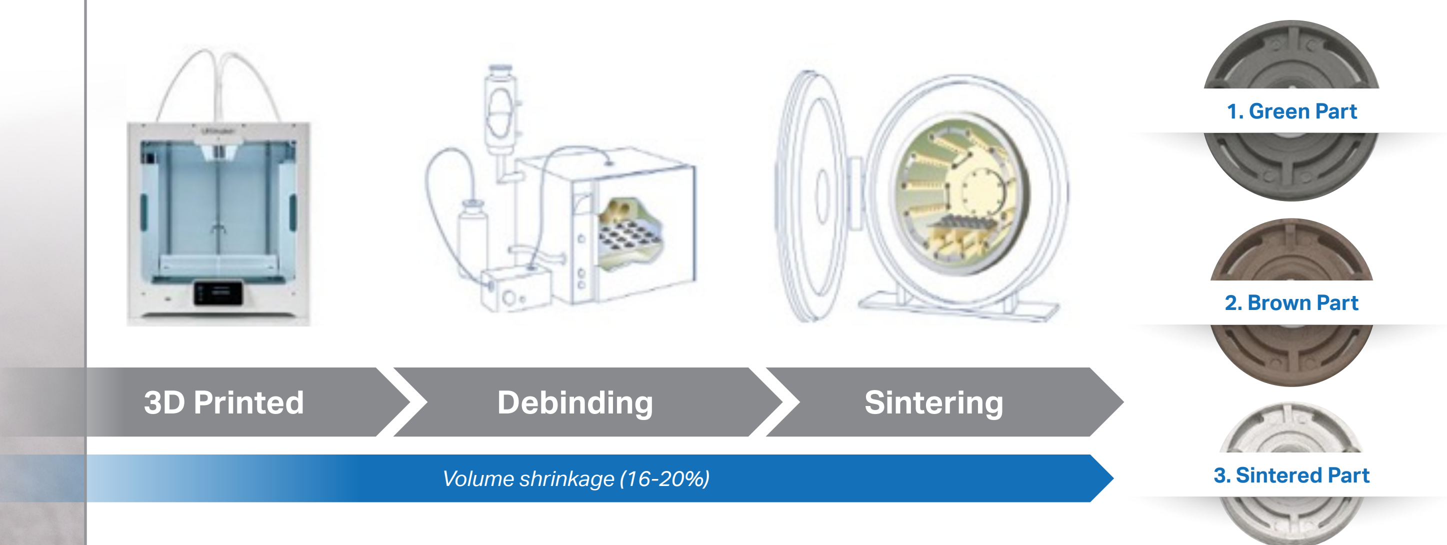
Different part geometries influence the degree of warpage. For example, flat, long parts are more prone to warpage compared to skinny, tall parts. Understanding the risk of warpage is essential during the design phase. Always discuss potential warpage and shrinkage issues with your vendor or print lab prior to printing.

Acknowledging and planning for shrinkage and warpage are key to successful 3D printing projects, ensuring the final parts meet the intended dimensions and accounting for these factors are crucial before starting your print.



Warpage Effects seen in 3D Printing

Shrinkage of a sinter-based metal 3D Printed Part



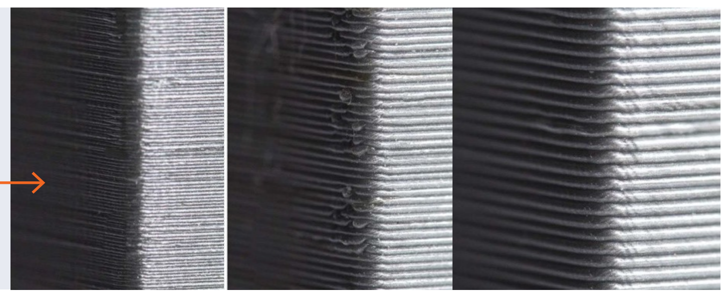
Resolution and Surface Roughness

In 3D printing, understanding resolution and surface finish is as crucial as in conventional manufacturing processes, with variations depending on the machine and technology used.

The choice of machine and AM technology significantly impacts the surface finish of parts. Moreover, given a chosen technology and machine, since parts are built layer by layer, the choice of how thick each layer is built (called layer thickness or layer height) also determines surface finish, resulting in varying surface textures, as illustrated in the provided images. Surface inaccuracies due to visible layer transitions are called layer lines.

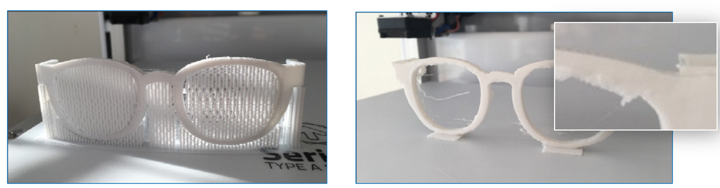
Machine Resolution and Support Surface Conditions

Machine resolution from varying layer thickness.



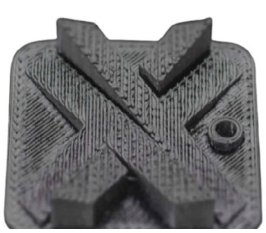
The placement of support structures can also lead to residual defects post-removal.

Besides machine setting and support cleanup challenges, feedstock and resulting surface finish from chosen material feedstock and technology should also be considered. At a high level, there are 3 main types of feedstock: **Filament**, **Resin** and **Powder**.



Example: *Notice how these glasses generated noticeable defects at points where support structures were removed, necessitating further cleanup.*

Image Source: medium.com



Filament feedstock produces visible layer lines as a characteristic of this process.

Resin-Based processes yield smoother surface finishes and produce the smoothest surface finishes right off the printer out of all 3 feedstocks.

Powder-based feedstock also has an absence of visible layer lines but results in a rougher, grainier texture. This texture can be polished using conventional methods as seen in the photo.

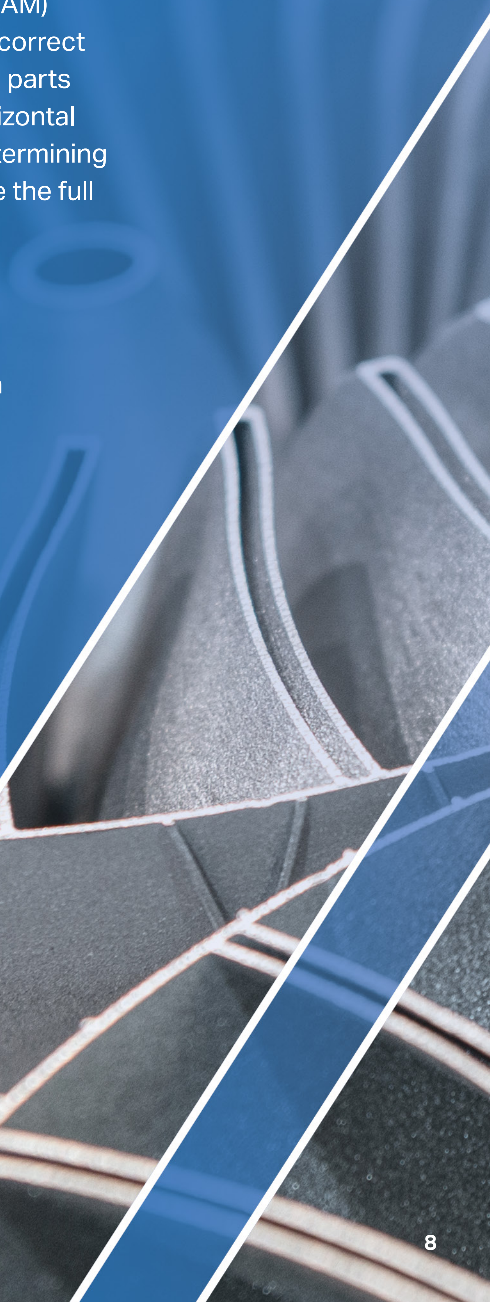
Image Source: Xometry

Image Source: eplus3d.com

CHAPTER 1

DfAM & Technical Considerations

In this chapter, we explore the essential design principles for additive manufacturing (AM) to achieve optimal performance and quality. Key areas of focus include selecting the correct AM process, enhancing part functionality, reducing material usage, and consolidating parts to streamline production. We will delve into specific guidelines such as managing horizontal overhangs, adhering to the 45-degree rule, ensuring minimum wall thickness, and determining appropriate hole and gap sizes. By mastering these concepts, designers can leverage the full potential of AM technologies, creating innovative, efficient, and reliable components.



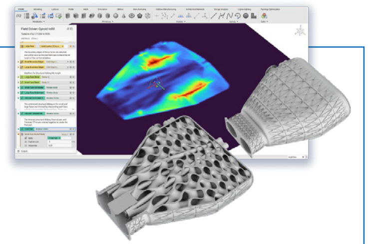
1.1 Designing for Additive Manufacturing (DfAM)

Exploring Opportunities in Designing for Additive Manufacturing (DfAM)

Designing for Additive Manufacturing (DfAM) offers vast opportunities for optimizing manufacturing processes. To fully leverage these benefits, consider four main aspects: **1.** Design for Correct AM Process and Finishing Steps, **2.** Design for Improved Part Functionality, **3.** Design for Material Reduction, and **4.** Design for Part Consolidation.

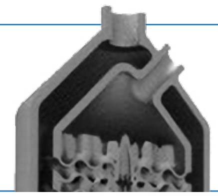
1 Design for the Correct AM Process & Finishing Steps

Each additive manufacturing technology comes with its own set of guidelines that must be followed to achieve the desired quality and functionality. Understanding these rules is essential to ensure the end product meets application-specific needs.



2 Design for Improved Part Functionality

Leveraging the capabilities of AM allows for enhanced part performance. This can involve increasing complexity without a cost penalty or improving mechanical properties through strategic design decisions.



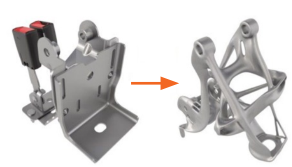
3 Design for Material Reduction

AM enables the creation of lighter, more efficient parts without compromising strength. Intelligent design can reduce material usage and overall weight, which is particularly beneficial in industries like aerospace and automotive.

4 Design for Part Consolidation

Merging multiple parts into a single, complex component minimizes assembly requirements, reduces the risk of failure, and potentially lowers production costs.

DfAM is about rethinking design to exploit the full potential of additive manufacturing. This approach can lead to more sustainable manufacturing processes, reduced lead times, and innovations not possible with traditional manufacturing methods.



8 Components into 1 Part

40% Lighter | 20% Stronger

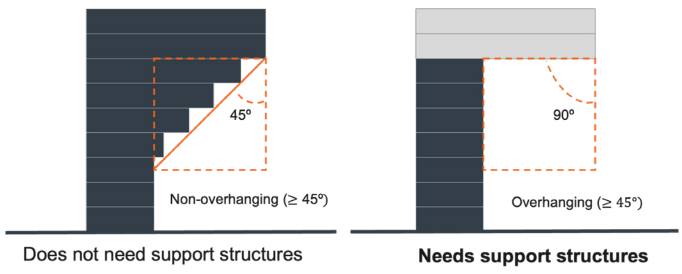
In addition to these general considerations, there are specific technical guidelines to account for when designing parts for 3D printing. The following sections will introduce concepts helpful across various AM technologies, highlighting technology-specific guidelines while delving into each AM technology in detail.

Horizontal Overhangs & the 45 Degree Rule Overview

Understanding Horizontal Overhangs and the 45-Degree Rule in 3D Printing

Horizontal Overhangs are sections of a 3D print that project outwards without any underlying support, often leading to defects such as sagging. Successfully printing these overhangs without sagging depends largely on the material selected and the specific 3D printing process employed.

The feasibility of printing overhangs **without sagging** varies based on the material used and the specific 3D printing process.



The 45-Degree Rule

As a general guideline, overhangs should be designed with angles **greater than 45 degrees** from the build plate. This helps avoid the need for support structures and mitigates defects like curling, where the edges of the part lift up from the build plate.

Image Source: crealitycloud.com

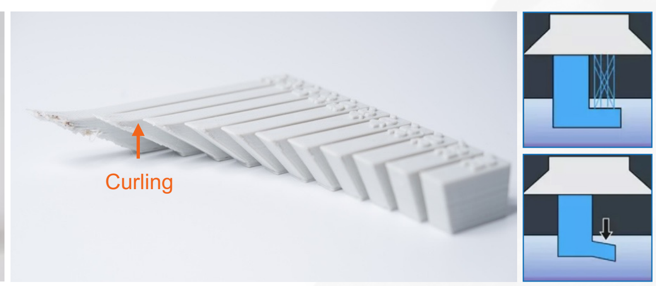


Image Source: medium.com

Material and Process Considerations

Material Selection: The material used can significantly impact the ability to print overhangs without defects.
Printing Process: Some printing technologies may allow for greater overhang angles. For example, Selective Laser Sintering (SLS) may not require support structures at all.

Incorporating these considerations when designing parts for 3D printing is crucial for ensuring structural integrity and print success.

Note: *This is a general rule and some technologies can achieve greater angles relative to plate and some technologies such as SLS do not require supports at all.*

1.2 Horizontal Overhangs

Overhangs refer to the parts of a 3D print that extend outwards, unsupported from below.

They're particularly challenging in 3D printing, especially when it comes to materials like polymers and metals. The feasibility of printing overhangs without deformations or drooping varies based on the material used and the specific 3D printing process.

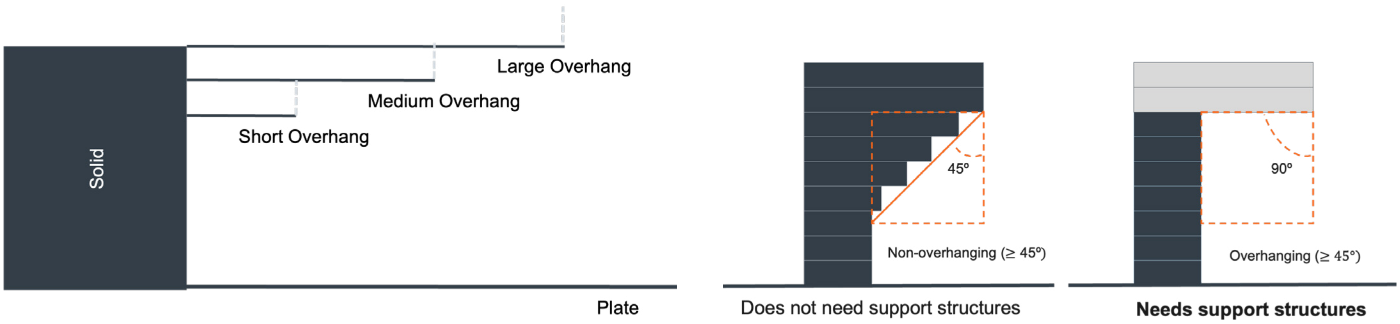


Image Source: Instrutables

CONSIDERATIONS:

Polymer vs. Metal: While both materials face challenges with overhangs, the physical properties of metals can sometimes allow for slightly larger overhangs without support, but with limitations.

Core Wall Thickness: In metal 3D printing, the thickness of the core wall plays a pivotal role. A thicker core wall allows for potentially larger overhangs. However, even with an optimal thickness, overhangs greater than 3mm are prone to deformation.

Material Extrusion: This method, commonly associated with FDM (Fused Deposition Modeling), allows for overhangs equivalent to 2-3 bead widths. Beyond this, the risk of deformation or drooping increases.

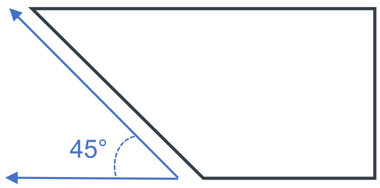
Support Structures: When printing complex geometries with significant overhangs, the use of support structures becomes essential. These structures provide the necessary support during printing and are removed post-printing.

Printing Angle: Adjusting the printing angle can mitigate some of the challenges associated with overhangs. An angle of 45 degrees or less is generally recommended for overhangs without supports.

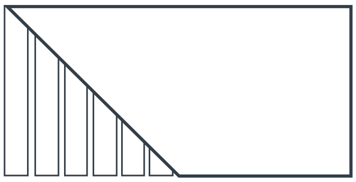
1.3 45 Degree "Rule"

In 3D printing, surfaces angled at 45° or less yield crisp results without needing auxiliary supports.

Beyond this angle, support becomes essential. Sticking to the 45° guideline ensures a smoother print and reduces post-processing effort.



Overhang of less than 45 degrees
No support is needed.



Overhang of more than 45 degrees
Support is needed.

Drooping

Cause: As layers are printed, the yet-to-harden plastic is tugged down by gravity, leading to droopy and elongated strands.

Result: The final print may exhibit sagging areas, especially in overhanging regions.

Curling

Cause: When the printed material doesn't cool down uniformly or swiftly enough, it tends to curl upwards.

Result: The print manifests a rugged, inconsistent finish, especially on its underbelly.

Image Source: hubs

The effect of increasing overhang angle (in increments of 5°) on print quality. The maximum angle shown is 70°.

Compromises to Acknowledge

Quality: Overhangs, when not addressed appropriately, can degrade the overall appearance and smoothness of the print.

Efficiency: The need for support can extend print times and increase material usage. Moreover, removing these supports post-printing can be a meticulous task.

Structural Integrity: Both drooping and curling can weaken the structure, making it less durable or stable in the long run.

1.4 Vertical Walls: Minimum Thickness

In traditional manufacturing, parts typically have a uniform structure.

In contrast, 3D printing distinctly separates parts into interior (infill) and exterior (shell) portions. The infill can vary in density and structure, while the shell remains solid, both influencing the mechanical properties of the print. The shell comprises vertical walls and horizontal top and bottom layers, defined by their orientation; walls rise vertically, while top and bottom layers span horizontally.

In the realm of 3D printing and additive manufacturing, the orientation and thickness of walls play pivotal roles in the success of a print. Particularly, vertical walls have their own unique considerations.

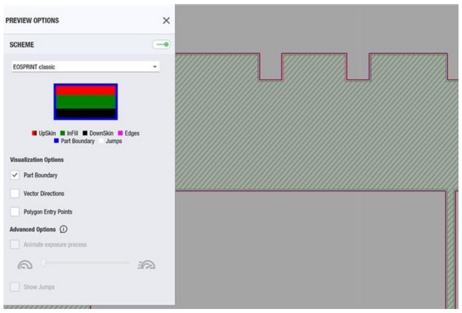
VERTICAL WALL CONSIDERATIONS:

Orientation Advantage: Vertical walls, being perpendicular to the print bed, often showcase better structural stability than their angled counterparts. This makes them conducive for achieving narrower profiles without compromising the integrity of the print.

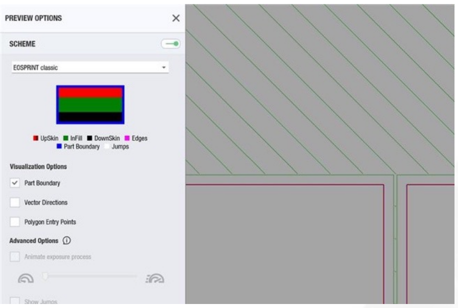
Thickness Threshold: While vertical walls can be made slimmer, it's crucial to maintain a minimum thickness. Features under 0.5mm are susceptible to inconsistencies or even print failures due to their delicate nature.

Engineer Consultation: For walls that tread in the borderline range, specifically those under 2mm, it's prudent to consult an additive manufacturing (AM) engineer. Their expertise can provide guidance on feasibility, necessary modifications, or support structures to ensure print success.

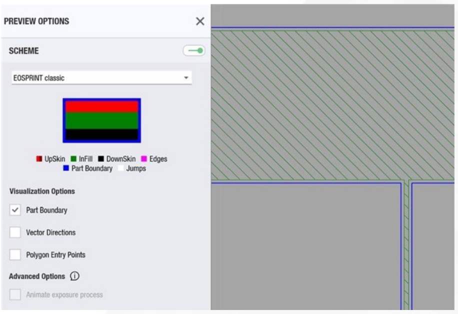
Post-Processing Concerns: Thinner walls, even if printed successfully, may pose challenges in post-processing. Their fragility could lead to breakages or deformities during cleaning or other finishing processes.



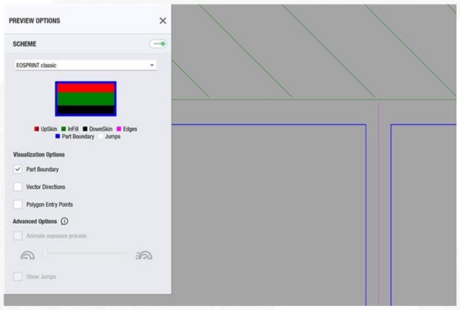
Thickest wall with the most Hatch/Infill & 1 Border/Contour



Thin wall with minimal Hatch/Infill and 1 Border/Contour



Solid wall with Hatch/Infill and 1 Border/Contour



Thinnest wall with only a single deposition line (often does not resolve or function)

1.4 Vertical Walls: Aspect Ratio

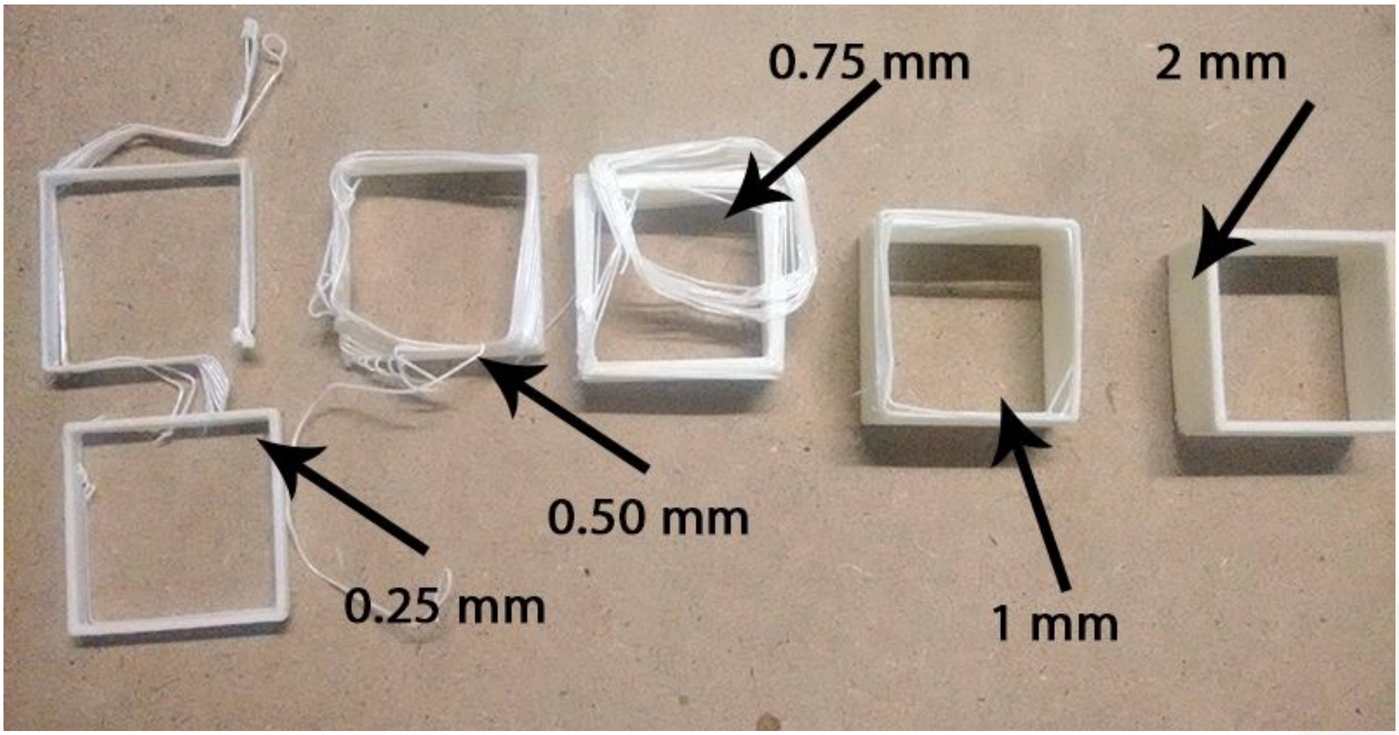


Image Source: fabacademy.com

The Importance of Aspect Ratio in Designing Vertical Walls for 3D Printing

Vertical walls have the capability to be designed slimmer than their angled counterparts.

For optimal results and structural integrity, it's advisable to avoid designing walls or features that are thinner than **0.5mm**.

If a design necessitates walls with a thickness of **less than 2mm**, make sure to have it evaluated by an experienced additive manufacturing engineer to ensure the best outcome.

Be aware that depending on the technology used, there are **height limitations** for walls over 2mm in thickness.

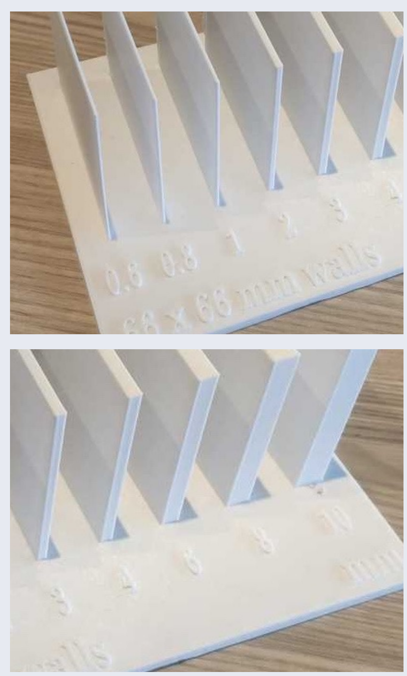


Image Source: cults3d.com

1.5 Holes & Gaps: Hole and Thread Design

Considerations for Holes and Threads in 3D Printed Components

When designing 3D printed components with holes and threads, specific considerations are essential to ensure functionality and quality. By incorporating these design strategies, the final printed parts can achieve both functionality and high quality. These solutions demonstrate that despite the limitations of current technology, effective methods exist to overcome common challenges in 3D printing.

HOLE DESIGN CONSIDERATIONS:

Horizontal Holes: Typically require internal support structures to maintain shape during printing.

Alternative Geometries: Designing holes with diamond or teardrop shapes can eliminate the need for supports, simplifying the design and reducing post-processing work.

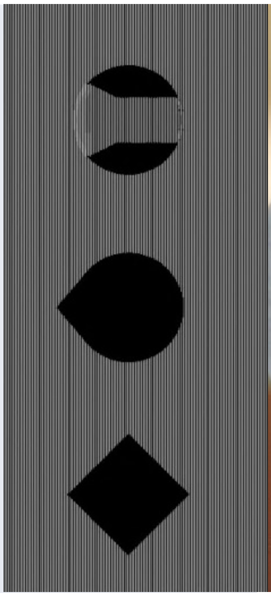
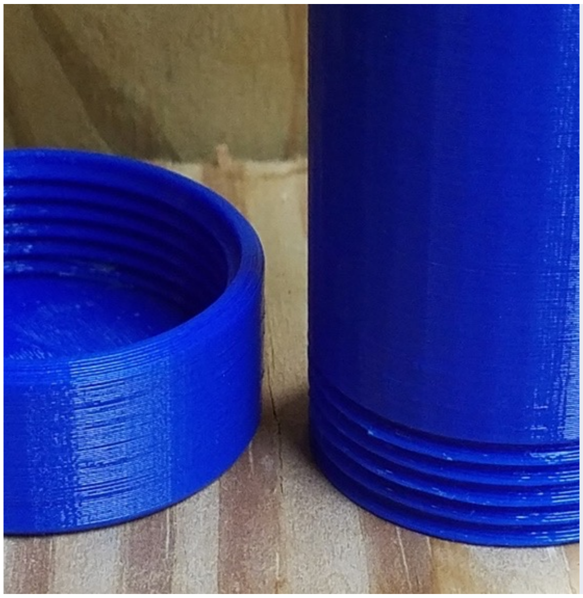


Image Source: hackaday.com



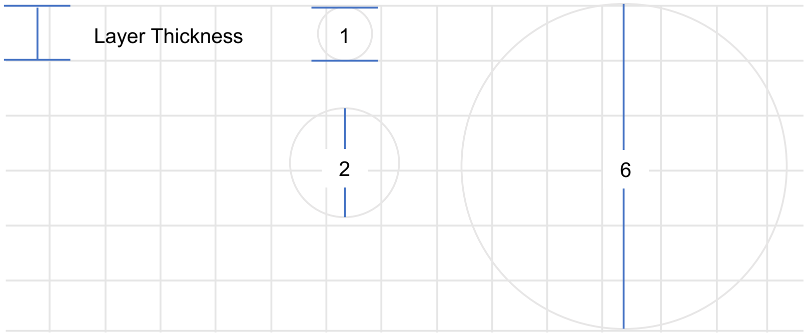
THREAD DESIGN CONSIDERATIONS:

3D Printed Threads: While functional threads can be printed, they may not match the precision of machined or injection-molded threads.

Threaded Inserts: For high precision or durability, incorporating threaded inserts into the 3D print provides robust threading capable of withstanding repeated use.

1.5 Holes & Gaps: Minimum Hole/Gap Size

The accuracy and quality of holes or gaps produced during additive manufacturing depend significantly on their orientation and size.



Vertical Holes:

Visibility Range: Generally visible within the 0.5-1mm size range.

Powder Intrusion Risk: Smaller holes may be susceptible to getting filled with leftover powder, compromising hole integrity.

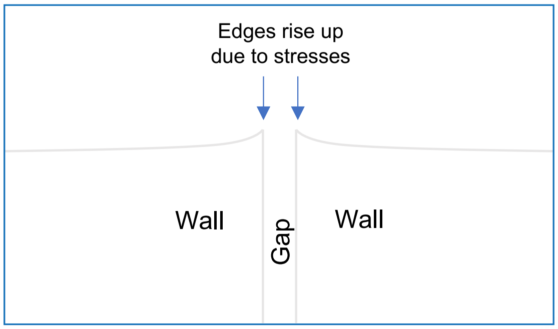
Horizontal Holes:

Visibility/Functionality Limits: When the diameter is too small (<4-5 layers typically) the holes may not be visible at all, but holes under 10-15 layers may come out at diamonds or just not be functional depending on the printing process and material.

Shape Retention Concern: When the diameter exceeds 8mm, holes might lose their characteristic round shape.

Build Failure: Significant deformities in horizontal holes can result in overall build failures, underscoring the importance of precise calibration and parameter maturity.

Potential Solution: Holes no longer need to be round when using AM, they can be a teardrop, diamond, or oval shape to minimize the topmost radius of the hole.



Functional Gaps

Adequate spacing between walls is essential to account for potential edge swelling and to ensure the removal of any powder or fibers. This space often depends on printing parameters. While adjustments can be made, the gap size typically correlates with the thickness of the surrounding part: thicker parts demand larger gaps. This principle also applies to holes constructed along the Z-axis.

Angled Holes:

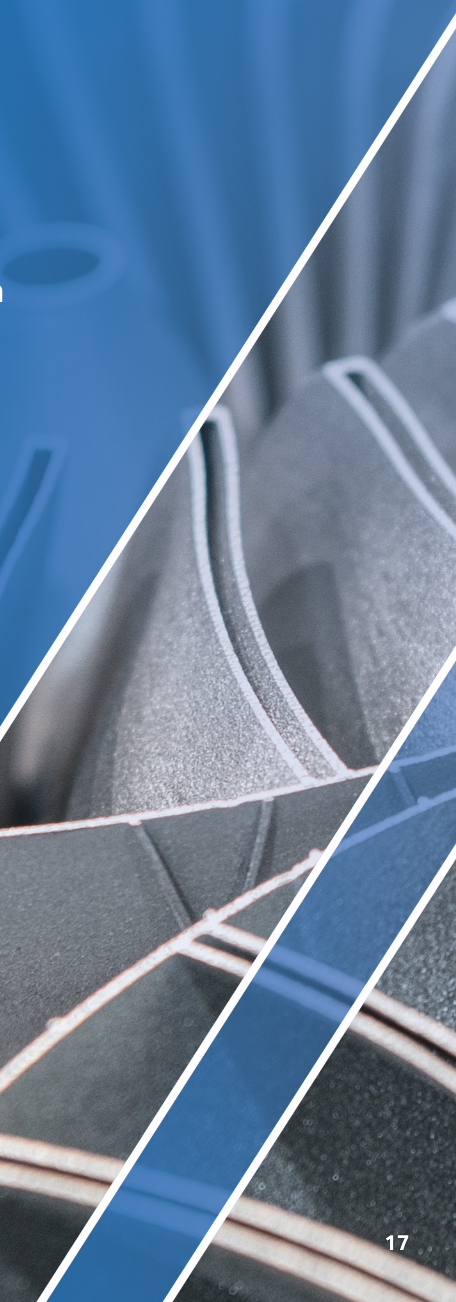
Elongation Tendency: Holes set at an angle, relative to the build platform, can experience elongation during the print, altering the intended design dimensions.

CHAPTER 2

Technology Deep Dive

This chapter provides a comprehensive overview of the various Additive Manufacturing (AM) technologies available today. From Material Extrusion and Vat Polymerization to Material Jetting, Binder Jetting, and Powder Bed Fusion, each method offers unique advantages and challenges.

We will delve into the specifics of each process, covering essential aspects such as material compatibility, build volumes, equipment manufacturers, and common failures. Understanding these technologies will help you choose the right method for your specific application, ensuring optimal results.



Technology Overview

Technical Specifications and Practical Considerations

Beyond technical specifications, this chapter explores practical considerations such as build size, layer resolution, surface finish, and post-processing requirements. By understanding these factors, designers can anticipate potential challenges and address them proactively during the design phase. For example, knowing that certain technologies may require extensive support structures can influence design decisions related to part orientation and geometry, ultimately reducing material waste and print time.

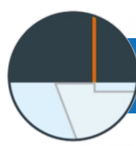
Navigating the Additive Manufacturing Landscape

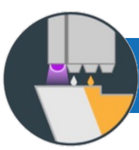
To effectively navigate the landscape of additive manufacturing, it is essential to grasp the nuances of different technologies. This knowledge enables designers to select the appropriate AM process for their specific application, balancing factors such as speed, cost, and material properties. Moreover, understanding the interplay between design and technology fosters innovation, allowing designers to push the boundaries of what is possible with AM, creating parts that are not only functional but also optimized for performance and manufacturability.

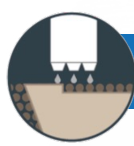
AM Processes Reshaping Manufacturing Landscapes

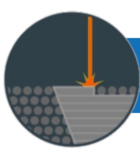
In this section, we will dive deep into the key additive manufacturing (AM) processes that are reshaping manufacturing landscapes. The technologies we will explore include:


Material Extrusion


Vat Polymerization

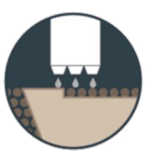
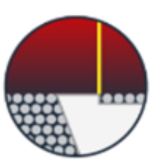
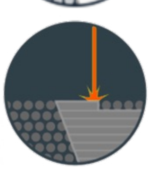

Material Jetting


Binder Jetting


Powder Bed Fusion

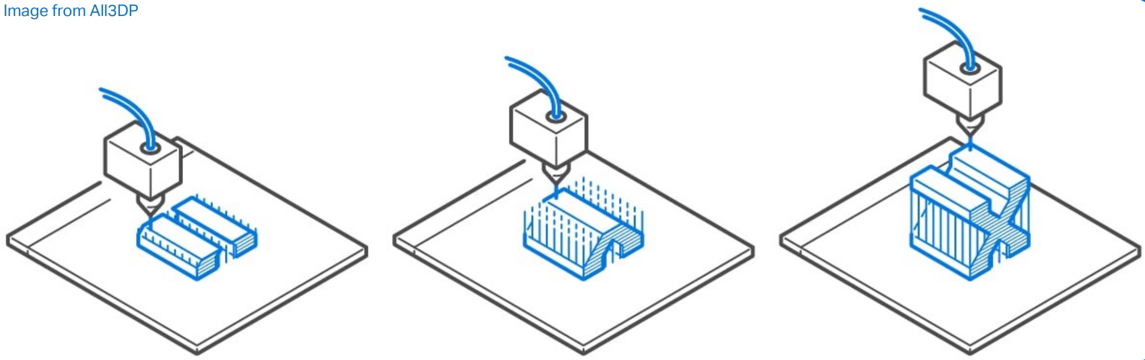
Each of these AM technologies offers unique flexibility and potential for innovation. As we examine these processes, consider not just how they work, but also why they are pivotal for achieving cutting-edge, efficient production in modern industries.

Technology Overview

Process Category	Process Overview	Technologies
 **Material Extrusion**	Material is selectively dispensed through a nozzle or orifice, which is typically heated to soften or melt the material.	• Fused Deposition Modeling (FDM) • Fused Filament Fabrication (FFF) • Bound Metal Deposition (BMD) • Atomic Diffusion Additive Manufacturing (ADAM) • Big Area Additive Manufacturing (BAAM)
 **Vat Polymerization**	Liquid photopolymer in a vat is selectively cured by light-activated photopolymerization.	• Stereolithography (SLA) • Digital Light Processing (DLP) • Gel Dispensing Printing (GDP)
 **Material Jetting**	Droplets of build material are selectively deposited.	• Polyjet • Multijet • Nanoparticle Jetting
 **Binder Jetting**	A liquid bonding agent is selectively deposited to bind powder materials, typically followed by sintering to densify the bound powder.	• Binder Jetting (BJ) • Powder Inkjet Printing • 3D Printing (3DP) • Digital Metal, Single-Pass Jetting (SPJ)
  **Powder Bed Fusion**	Focused thermal energy selectively fuses regions of a powder bed.	• Selective Laser Melting (SLM) • Direct Metal Laser Sintering (DMLS) • Selective Laser Sintering (SLS) • 3D Metal Printing (3DMP) • High-Speed Sintering (HSS) • Multi-Jet Fusion (MJF)

CHAPTER 2.1
Material Extrusion

Image from All3DP



Material extrusion is a process that creates 3D objects by pushing a material through a heated nozzle and laying it down in layers. The material can be a thermoplastic or a composite that melts or softens when heated and hardens or cures when cooled.

Feedstock type: Filament

Common technologies: FDM or FFF (Fused Deposition Modeling, Fused Filament Fabrication), material extrusion is the most widespread and prevalent form of AM.

Industrial Build Volumes

- Desktop – up to 400 x 400 x 300 mm
- Large Filament Fed– up to 500 mm3
- Pellet Fed (also known as Large Format) 1 m3 to 10 x 2 x 3 m

Equipment Manufacturers

- 3D Systems
- Stratasys
- Markforged
- Ultimaker
- Aon3D
- BigRep
- Juggerbot

Common Materials

- ABS, ASA
- PA (Nylon)
- PETG, PLA
- Polycarbonate (PC)
- PEEK, PEKK
- ULTEM
- Composites

Cost ($$$$)

Machines

Entry Level | < $1k

Prosumer | $500 - $10k

Industrial | $20k – 1M

MATERIAL EXTRUSION

Design Guidelines & Common Failure Modes

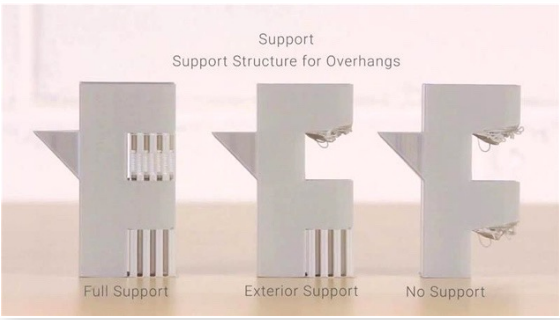
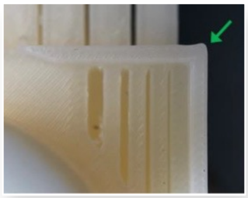
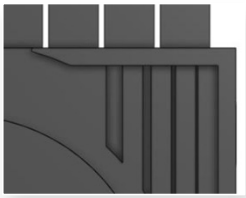


Image Source: Tinkerine

Print Orientation and Support

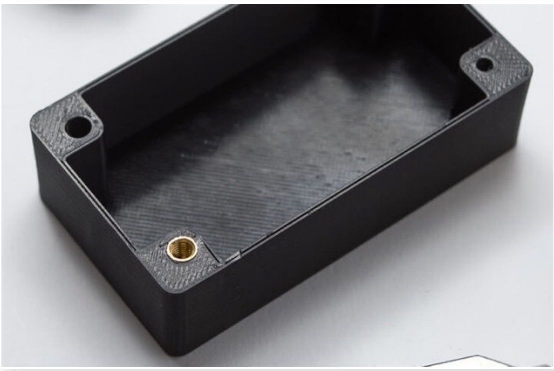
With FDM, it is critical to orient the part in a way that minimizes the amount of support material required as much as possible. Orientation is also key for strength, FDM parts will be strongest in the XY plane, and weakest in Z-direction, specifically in tension in the Z-direction.





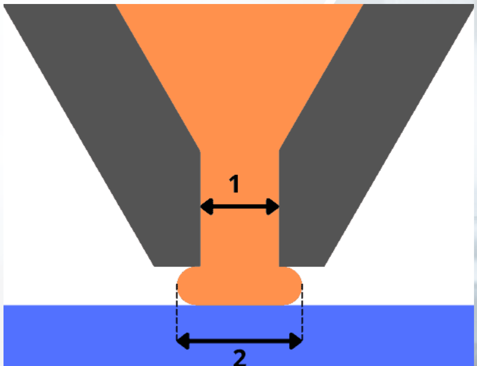
Edges and Corners

Typically FDM does not produce sharp corners and edges. “rounding” can occur give the nozzle type chosen. This should be accounted for in the design.



Drilling out holes and Inserts

Due to the overall lower print resolution with material extrusion machines, it is best practice to undersize all holes and drill them out to the proper size after printing. Threads may be printed, but it is best practice to use threaded inserts for accuracy.



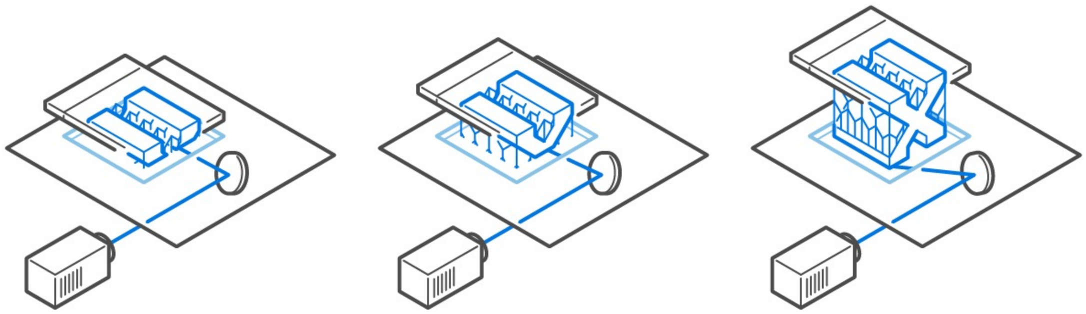
FDM Nozzle Size Comparison

Nozzle Size to Bead Width: Depending on nozzle size used, the bead width or minimum feature will generally be 2x the nozzle size.

CHAPTER 2.2

Vat Polymerization

Image from All3DP



Vat Photopolymerization printers include any technologies where liquid photopolymer in a vat is selectively cured by light-activated (laser or digital projector) photopolymerization. This category basically includes any printer using liquid as the main material for building a part.

Feedstock type: Liquid Resin

Common technologies: SLA (Stereolithography) or DLP (Digital Light Processing). "SLA" is the most common.

Industrial Build Volumes

- Desktop – up to 150 x 150 x 200 mm
- Professional – up to 300 mm3
- Industrial – up to 1 m3

Equipment Manufacturers

- Stratasys
- Formlabs
- Carbon
- 3D Systems
- Nexa3D
- Astra3D

Common Materials

- Photopolymer resins typically proprietary by machine manufacturer, in contrast to other technologies wear many materials are common
- Castable waxes
- Transparent
- Rigid / Tough
- Bio-compatible

Cost ($$$$)

Machines

Entry Level | < $1k

Prosumer | $5k - $10k

Industrial | $20k – 1M

VAT POLYMERIZATION

Design Guidelines & Common Failure Modes

Cross-sectional area: The larger the cross section, the higher the force to remove it from the film - requiring denser support structures. Longer delay times will also be needed to allow the material to flow.

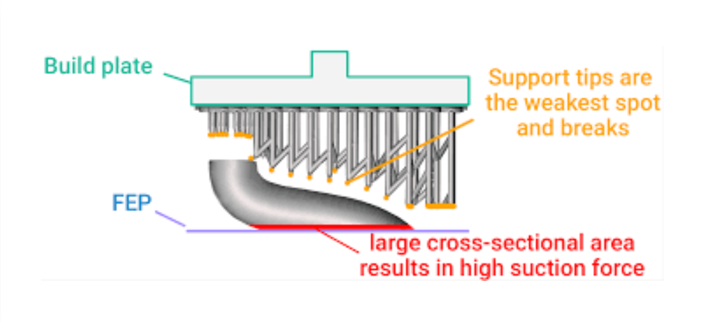


Image Source: ameralabs.com

Image Source: cubeek3d.com

Cupping: Some geometries can capture air between the resin surface and the film. Think of putting a cup underwater upside down – it traps the air. This pressure prevents the resin from flowing properly and can result in a failed print.

Drain / relief holes: If the part has internal voids, there will need to be holes modeled in the part to drain the resin. Typically, these types of geometries also result in cupping, so it is best practice to model the holes in such a way to relieve the pressure during the printing process.

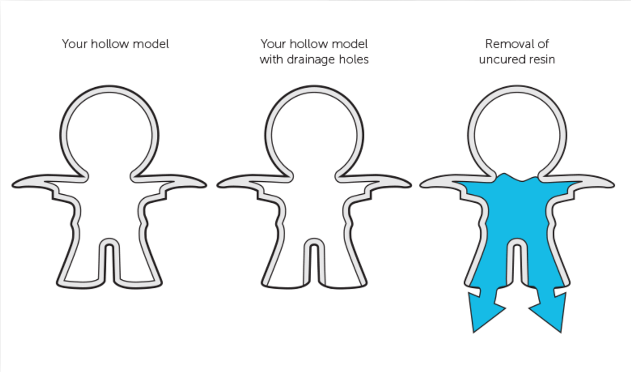
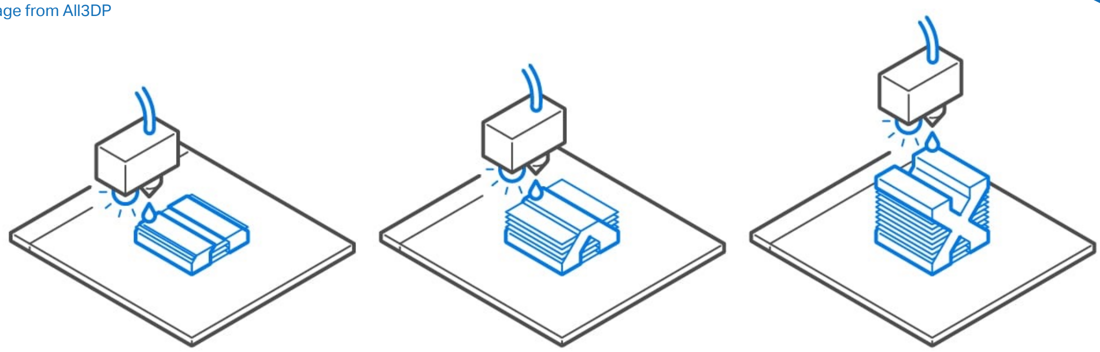


Image Source: materialise.com

CHAPTER 2.3

Material Jetting

Image from All3DP



Material Jetting (MJ) is a sophisticated 3D printing technique that operates much like traditional inkjet printing but in a 3D space. Commonly referred to as PolyJet it can produce multi-color prints and can create components that are flexible or rigid.

Feedstock type: Liquid Resin

Common technologies: Polyjet, Nano Jetting, Drop-on-demand

Industrial Build Volumes

- **Professional:** Up to 100 x 100 x 50 mm
- **Industrial:** Up to 508 x 508 x 305 mm

Equipment Manufacturers

- Stratasys
- HP

Common Materials

- Proprietary photopolymers
- Standard / Rigid (PE)
- ABS-like
- PP-like
- High Temp
- Castable
- Biocompatible

Cost ($$$$)

Machines

Professional | $50k - $100k

Industrial | $100k – 1M

MATERIAL JETTING

Design Guidelines & Common Failure Modes

Mechanical Strength: Material jetted parts, lacking nylon or ABS, are weaker and more brittle due to acrylic resin. Their low heat tolerance and minimal elongation in rubber-like materials limit their use in functional testing and real-world applications.

Glossy vs. matte: The matte setting will add a thin layer of support across the entire part, regardless of orientation or requirement. The glossy setting will only use support material where required to allow for the building of the model.

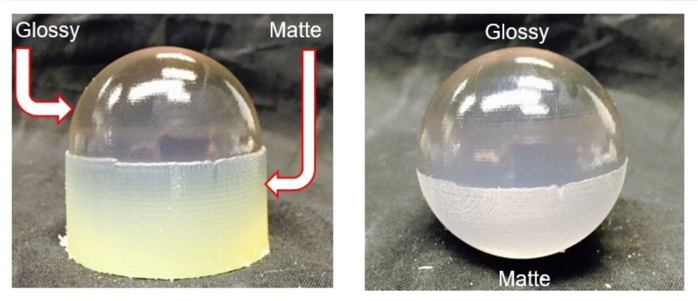


Image Source: grabcad.com

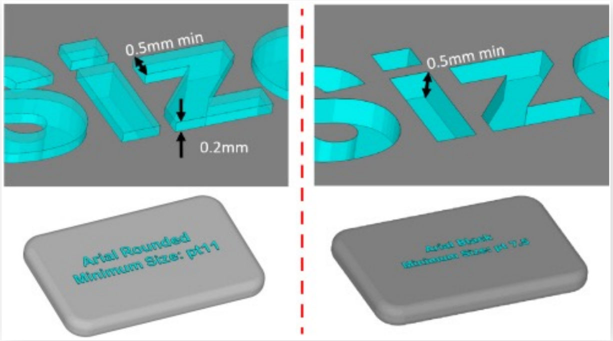


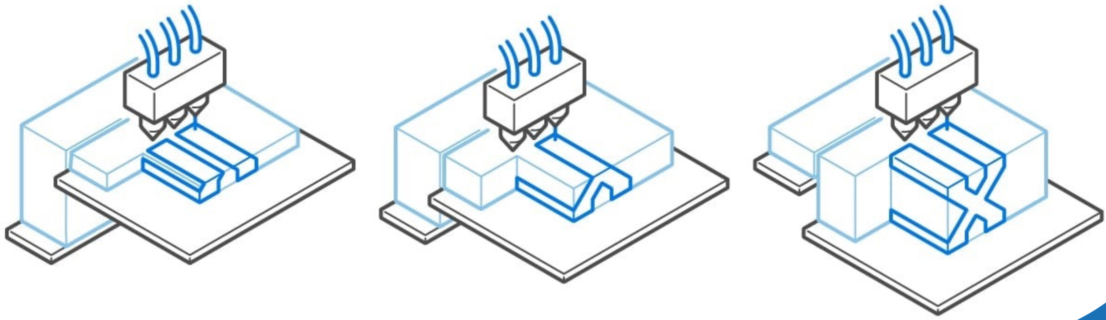
Image Source: stratasys.com

Embossed and engraved details, walls, holes, and pins - To ensure small details are visible and features are successful, these should all be a minimum 0.5 mm in depth / thickness / diameter.

CHAPTER 2.4

Binder Jetting

Image from All3DP



Binder Jetting is a 3D printing process where a liquid bonding agent selectively binds regions of a layer of powder. The technology uses a powder material (metal, plastic, ceramic, wood, sugar, etc.) and a liquid material deposited from inkjets. Printers are typically built to work with one type of material ex: metals, plastics, ceramics, etc.

Feedstock type: Powder

Common technologies: Binder Jetting, or Drop-on-powder printing

---

Industrial Build Volumes

- Development Sizes up to 100x200x100mm
- Production metal systems up to 1x1x1m
- Sand systems up to 2x2x1m
- Polymer systems up to 380x380x380mm

Equipment Manufacturers

- HP
- ExOne
- Desktop Metal
- GE
- Rapidia

Common Materials

- 316L SS
- Maraging Steel
- 17-4PH SS
- Nickel 625
- Nickel 718
- Copper
- Sand
- Nylon

Cost ($$$$)

Machines

Entry Level | -

Prosumer | -

Industrial | $50k – 1M

BINDER JETTING

Design Guidelines & Common Failure Modes

Image Source: Colibrium Additive

Geometric Freedom: Binder jetting has nearly complete geometric freedom; it can create parts with complex internal channels, lattices, sharp edges and corners, and encapsulated mechanisms.

Design for Shrinkage: Sintering shrinks the volume of the part by 15 to 16%, but the shrinkage is predictable, repeatable, and can be modeled in software, allowing for fine dimensional precision despite the size change.

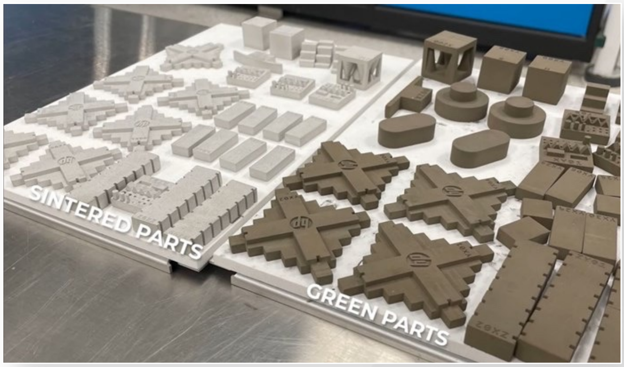


Image Source: HP

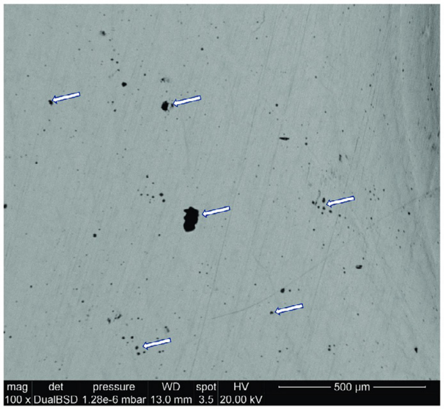
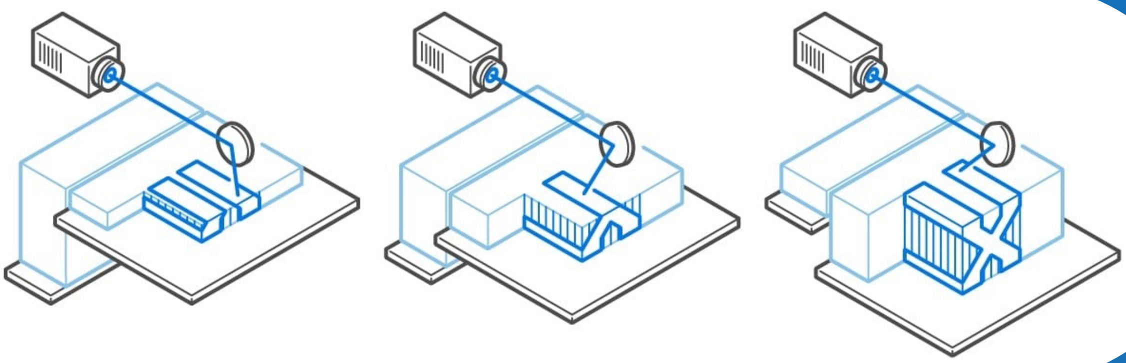


Image Source: mdpi.com

Porosity: Due to the shrinkage occurring form green part to final sintered part, there typically occurs a high degree of porosity in the final sintered part. This should be accounted for in high critical applications and industries such as Aerospace and Medical.

CHAPTER 2.5

Powder Bed Fusion



Powder bed fusion (PBF) is a 3D printing process where a thermal energy source such as a laser selectively melts powder particles inside a build area to create a solid object layer by layer. It has reached widespread usage in the aerospace and defenese sectors due to its ability to create high strength unique geometries.

Feedstock type: Powder (Metal or Polymer)

Common technologies: DMLS (Direct Metal Laser Sintering), SLM (Selective Laser Melting), SLS (Selective Laser Sintering), LMF (Laser Metal Fusion), and EBM (Electron Beam Melting)

Industrial Build Volumes

- Small – 100-200W Laser Up to 150x150x150mm
- Medium – 200-500W Laser Up to 350x350x400mm
- Large – 500-1200W Lasers Up to 600x600x1000mm

Equipment Manufacturers

- EOS
- Velo3D
- GE Additive
- SLM Solutions
- 3D Systems
- Renishaw
- TRUMPF
- DMG Mori
- AddUp

Common Materials

(Metal)

- 316L SS
- Maraging Steel
- 17-4PH SS
- Nickel 625
- Ti64
- AlSi10Mg
- AL6061RAM
- Haynes 282
- Copper
- Cobalt Chrome

(Polymer)

- Nylon
- PEEK
- PEKK
- TPU
- TPE
- Polypropylene

Cost ($$$$)

Machines

Entry Level | ~$30k

Industrial | $20k – 2M+

POWDER BED FUSION

Design Guidelines & Common Failure Modes

Preventing Warping: typically long flat parts greater than 6" in length are prone to warping. It's best to consult with an experienced AM engineering to understand how parts can warp.

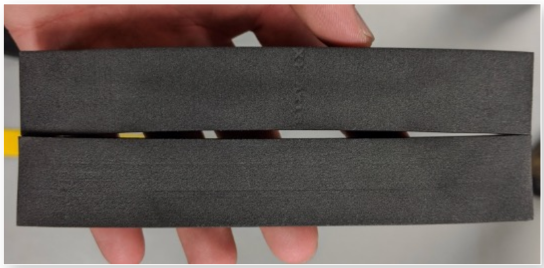
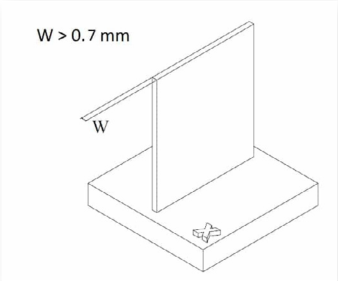


Image Source: formlabs.com



Feature Resolution: Features should be designed greater than 0.7mm.

Powder Removal Design: Account for ease of post-processing. SLS requires powder to be removed from all around the part.

Image Source: 3dprinting.com

Image Source: hubs.com

Surface Finish Post-Processing: Plan for necessary smoothing or dyeing techniques.

MENU

POWDER BED FUSION

Design Guidelines & Common Failure Modes

Overhangs & Supports ("anchors"): Required to mitigate warping and to anchor the part to the build platform, supports must be strategically placed and are later removed, which can influence the surface finish.

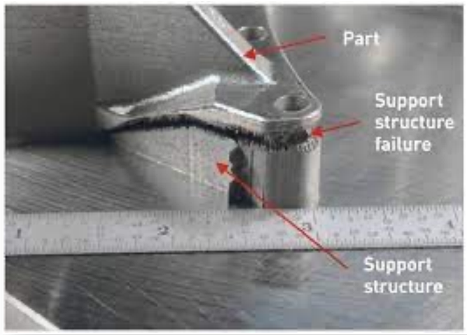


Image Source: metal-am.com

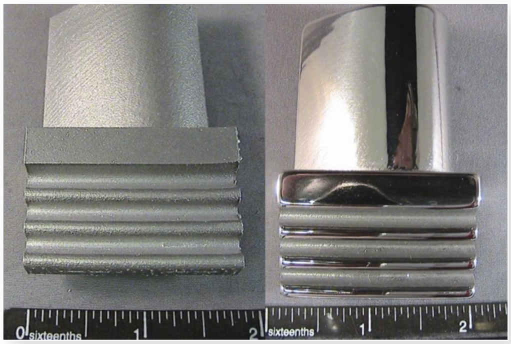


Image Source: tctmagazine.com

Surface Finish: As-built parts often have a rough surface that may require post-processing, depending on application requirements.

Residual Stress: Causing warping or cracking from uneven cooling.

Image Source: mobilityengineeringtech.com

CHAPTER 3

Designing for Additive Manufacturing 201

Advanced Design Strategies in Additive Manufacturing

Building on the basics, this chapter delves into advanced design considerations for additive manufacturing guiding you through the nuanced aspects of optimizing designs specifically for additive processes.

Key topics include:

- **Topology Optimization:** Utilizing computational tools to streamline material usage.
- **Thin-Walled Structures and Hollow Infills:** Enhancing part performance while reducing material and print time.
- **Strategic Part Placement:** Maximizing efficiency by placing parts strategically on the print bed.
- **Part Consolidation:** Combining multi-component assemblies and eliminating fasteners through innovative design.
- **Balancing Design Specifications with AM Limitations:** Exploring complex geometries, material optimization, part consolidation, and customization.
- **Iterative Design and Toolless Production:** Streamlining workflows and enhancing final products.

By understanding and applying these advanced guidelines, you can push the boundaries of what's possible with additive manufacturing, achieving superior results in both functionality and efficiency.

Designing for Additive Manufacturing 201

DfAM, or Design for Additive Manufacturing, is a comprehensive approach to designing components specifically intended for production using additive manufacturing (AM) technologies. Here's a breakdown:

1	**Complex Geometries**	DfAM allows for the design of intricate structures that would be challenging or impossible to produce with conventional methods. This includes lattice structures, internal channels, and organic shapes.
2	**Material Optimization**	Through DfAM, parts can be designed to use only the exact amount of material necessary, potentially reducing waste or allowing for a better material to be used.
3	**Part Consolidation**	Multiple parts in an assembly can often be combined into a single 3D-printed component, simplifying assembly and potentially improving performance, weight, and costs.
4	**Customization**	DfAM can cater to individualized designs, perfect for applications like medical implants tailored to a specific patient or bespoke consumer products.
5	**Iterative Design**	Rapid prototyping is a hallmark of AM. With DfAM, designers can swiftly iterate and improve upon designs, testing, and refining in real time.
6	**Toolless Production**	No molds or tooling are required in AM to print the part, granting designers more freedom and reducing lead times. Tooling may be required for post processing steps but can potentially be built into the AM part to use existing tooling.

In essence, DfAM is not just about making a design printable. It's about harnessing the full potential of additive manufacturing to achieve innovative, efficient, and tailored solutions. Adopting a DfAM mindset can lead to breakthroughs in design, performance, and cost efficiency. It is important to keep post processing steps in mind and incorporate them into the DfAM process.

Design for the Correct AM Process and Finishing Steps

Striking a balance between design specifications and the limitations of additive manufacturing is crucial. Knowledge of these nuances enables better design strategies, ensuring optimal results. Here are some design considerations:

Understanding Your Material

Different AM processes are suited for various materials - always consider the strength, flexibility, and thermal properties required for your part.

Layer-by-Layer Considerations

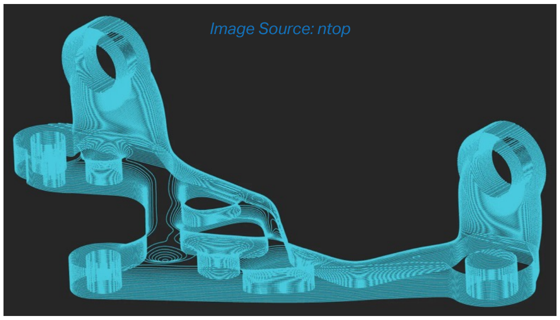
Orientation: The position of the part on the build platform can affect mechanical strength, printing speed, and the amount of required support structures.

Support Structures: Overhangs or areas without direct support below may necessitate temporary structures, which can impact print time, post-processing efforts, and material usage.

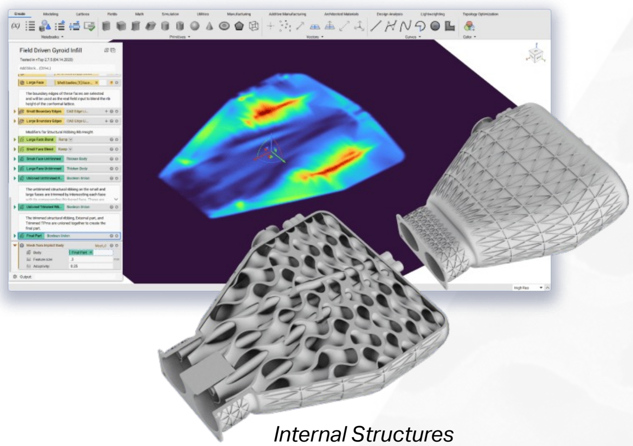
Layer Height: Choosing a smaller layer height can lead to smoother surfaces but will often increase print duration. Conversely, larger layer heights speed up printing but typically reduce detail and finish quality along the height (Z axis) of the part.

Internal Structures: Given the layer-by-layer approach, AM offers the freedom to createinternal lattices or hollow spaces that can't be achieved with other manufacturing methods, but in powder-based technologies, these volumes need to be accessible for powder removal.

Thermal Distortions: As each layer is deposited and solidified, it can induce thermal stresses which might warp the part, especially in designs with uneven geometries. This can be amplified further from the heated platform.



Layer by Layer Visualization



Internal Structures

Post-Processing Needs

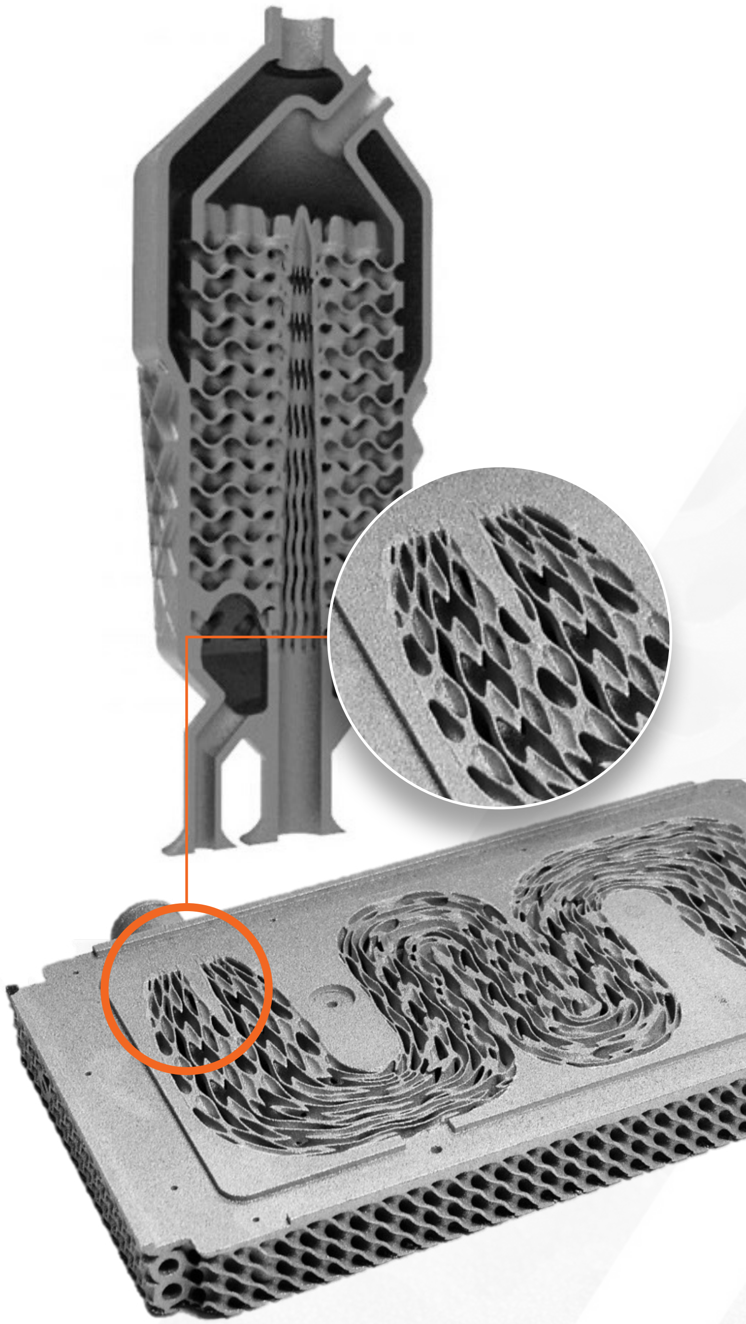
Consider how your part will be cleaned, refined, or finished post-print.

Production Volume

For prototyping or limited runs, processes like SLA or SLS (FFF? For small numbers, SLS and MJF can be packed in Z allowing for more parts in the build volume) might be ideal.

For higher volumes, consider processes with faster build rates, multi-part builds, or more printhead/lasers.

Design for Improved Part Functionality



Optimized Geometries:

Brief: Leverage AM's freedom to create intricate shapes, bypassing traditional manufacturing constraints.

Benefit: Enhanced aerodynamics, fluid dynamics, and overall performance.

Internal Lattices & Structures:

Brief: Craft lightweight, yet strong structures with unique internal geometries.

Benefit: Reduced weight without compromising on strength; improved material efficiency.

Integrated Assemblies:

Brief: Minimize assembly requirements by printing multi-component parts as a single unit.

Benefit: Decreased assembly time, reduced part count and minimized failure points.

Customized Surfaces:

Brief: Tailor surface textures and roughness for specific applications, from improved grip to reduced drag.

Benefit: Enhanced tactile feedback, improved aesthetics, and functional benefits.

Material Gradients & Composites:

Brief: Utilize varying material properties within a single print, offering regions of flexibility or rigidity as needed (in some technologies or by using multiple technologies collaboratively).

Benefit: Multi-functional components that cater to diverse application needs.

Design for Material Reduction

Topology Optimization:

Brief: Use computational tools to redesign parts, removing material from non-critical areas and only adding material in the required areas.

Benefit: Efficient material distribution, maintaining functionality while using less.

Thin-walled Designs:

Brief: Opt for slimmer walls without compromising part integrity, especially viable with AM's precision.

Benefit: Substantial material savings, faster print times, and balanced performance.

Hollow Infills:

Brief: Integrate customizable internal infill patterns, moving away from solid structures and only printing material required to support any overhanging features.

Benefit: Reduction in weight, material usage, and print time, while maintaining structural robustness.

Smart Orientation & Nesting:

Brief: Position parts intelligently on the print bed, allowing for maximal part production with minimal waste.

Benefit: Efficient use of print space, reduced supports, and material optimization.

Design for Part Consolidation

Reduce Assembly Efforts:

Brief: Transform multi-component assemblies into singular printed parts.

Benefit: Decrease in assembly time, labor costs, and potential failure points.

Enhanced Functional Integration:

Brief: Merge multiple functions (e.g., mechanical, thermal, fluidic) into a single component.

Benefit: Optimized performance with fewer parts, leading to enhanced reliability.

Eliminate Fasteners & Welds:

Brief: By consolidating designs, eliminate the need for screws, bolts, and welds.

Benefit: Reduction in weight, cost, and potential areas of weakness or failure.

Complex Geometries Made Possible:

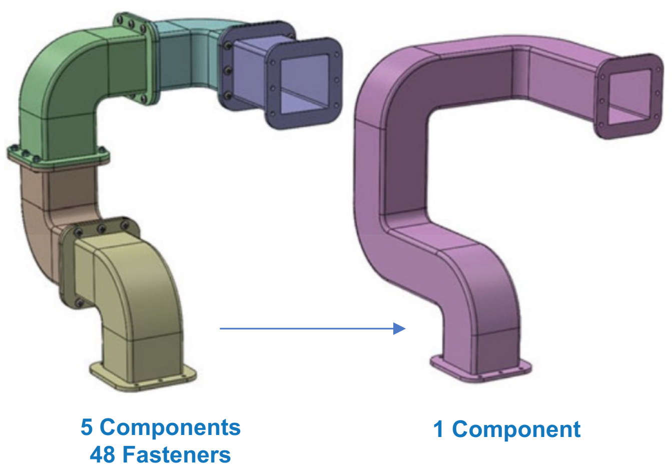
Brief: Leverage AM's capability to produce complex shapes, allowing the merging of traditionally separate parts.

Benefit: Greater design freedom and the potential for innovative solutions.

Inventory Reduction:

Brief: Fewer unique parts mean reduced inventory management complexities.

Benefit: Streamlined production, reduced storage needs, and lower carrying costs.



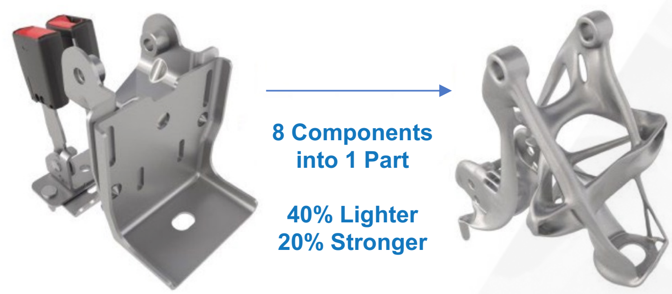
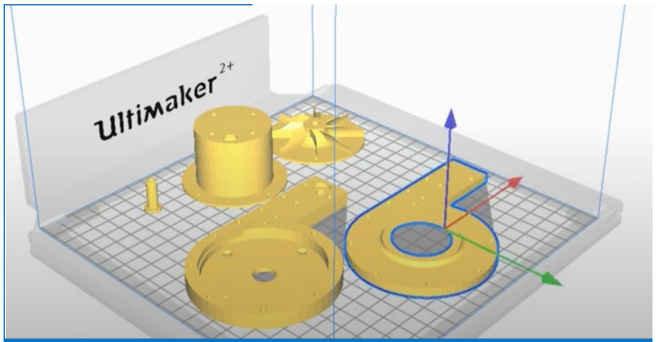


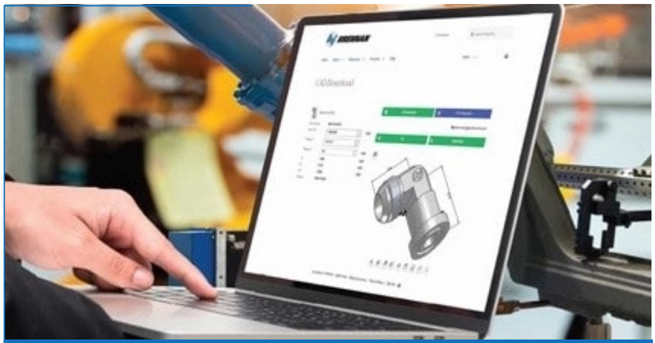
Image Source: kit

Speed Benefits



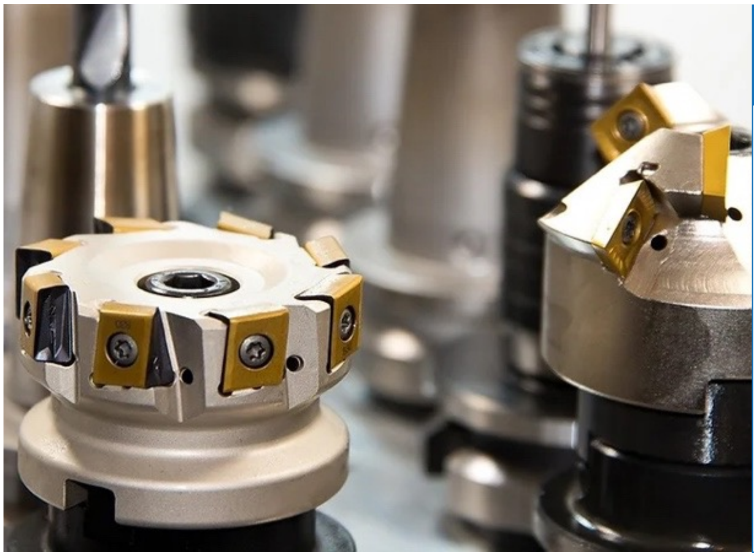
Multi-Design Printing

- **Batch Efficiency:** Print multiple variations of a design or entirely different parts in a single print cycle.
- **Parallel Production:** Achieve higher throughput by fabricating different components at once.
- **Time Savings:** Condense production timelines by avoiding sequential printing of individual parts.



Fast Cycle Innovation

- **Real-Time Testing:** Produce prototypes that can be immediately tested for form, fit, and function.
- **Accelerated Design Iteration:** Quickly adapt and reprint revised designs, shortening the development cycle.
- **Validation Speed:** Reduce time-to-market by rapidly validating design specifications and usability.



No-Tooling Required

- **Zero Setup Time:** Skip the delays associated with creating molds, jigs, or fixtures.
- **Agile Manufacturing:** Quickly switch between different designs without retooling.
- **Cost & Time Efficiency:** Eliminate the capital and lead time usually needed for tool production.

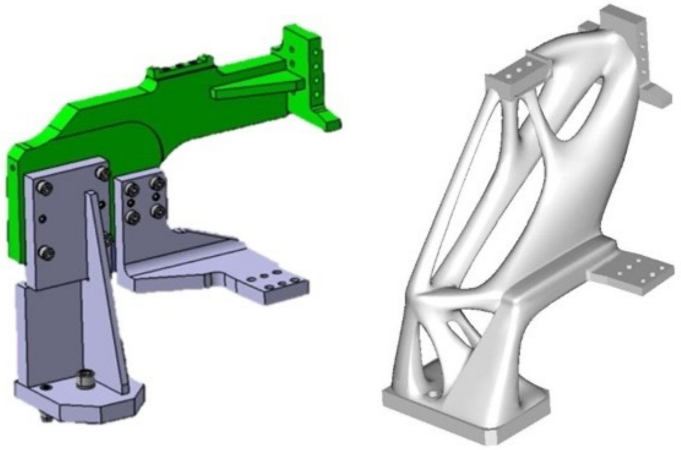
Cost Benefits

Mass Reduction

- **Material Efficiency:** Additive manufacturing allows for precise material usage, reducing waste.
- **Lightweight Components:** Tailor-made designs can achieve the same strength with less material, reducing overall mass.
- **Shipping Savings:** Lighter parts translate to lower transportation costs.

Design Consolidation

- **Complexity Without Cost:** Merge multiple parts into a single, intricate design without added costs for complexity.
- **Reduced Assembly:** Fewer components mean less time and cost spent on assembly processes.
- **Inventory Reduction:** Consolidate parts to reduce stockkeeping units (SKUs), simplifying inventory management.



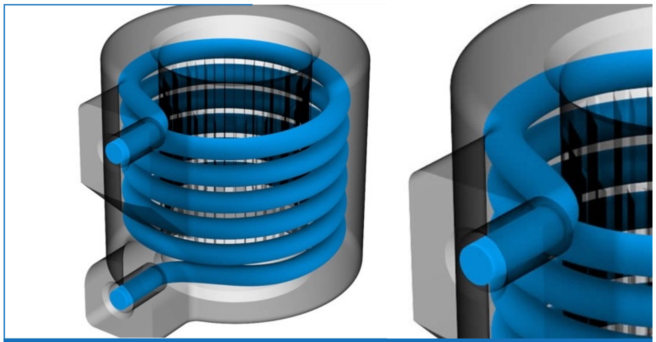
Overall Cost Optimization

- **Rapid Prototyping:** Quicker design-to-product cycle reduces R&D costs.
- **On-Demand Production:** Eliminate or reduce the need for warehousing by printing parts as needed.
- **Economics of Scale:** Easily adjust production volume without significant changes to the initial setup, providing cost flexibility. (Small to medium volume production.)

Performance Benefits

Design Freedom

- **Complex Structures:** Achieve intricate designs that are impossible or prohibitively expensive with traditional methods.
- **Overcome Limitations:** Bypass constraints associated with conventional fabrication techniques.
- **Integrated Features:** Produce parts with embedded features, reducing post-production steps.
- **Customization:** Easily adapt designs for niche applications or personalized end-user requirements.



Functional Performance

- **Optimized Geometry:** Ability to fabricate parts with optimized structures, enhancing strength and functionality.
- **Unlock Material Selection:** AM handles materials traditionally seen as challenging, often crafting superior parts. Moreover, AM offers a diverse range of alloys and their unique combinations, outpacing conventional techniques.
- **Custom Tailoring:** Ability to produce components specifically tailored to their end-use environment.
- **Enhanced Durability:** Integration of unique geometries can lead to longer-lasting components.

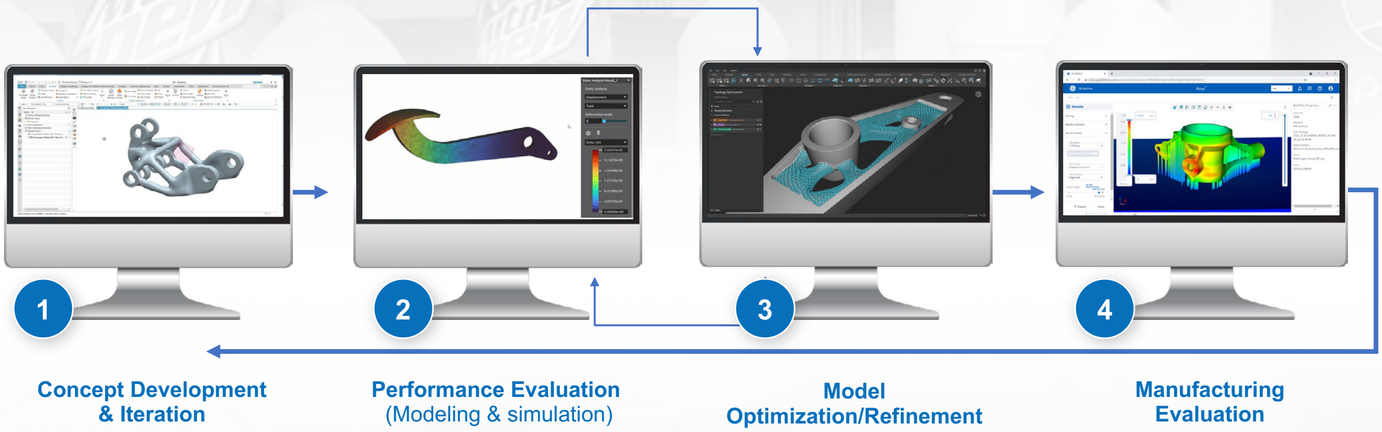
CHAPTER 4

Software and Tools to Assist in Design

In chapter 4 we focus on the software and tools that can assist in designing for additive manufacturing. We'll highlight our innovative software, ADDCAAM, powered by ADDMAN, and how it can revolutionize your design process.

Additionally, we'll cover other essential tools and techniques, such as topology optimization, thin-walled designs, hollow infills, and smart orientation. These tools not only enhance the precision and efficiency of your designs but also help you reduce material usage and improve overall performance. Embrace these technologies to stay ahead in the rapidly evolving field of additive manufacturing.

Software and Tools to Assist in Design



Prototyping


Image Source: PepsiCo

Prototyping is used as an aid during every step of the process progressing from fit/form to fully functional prototypes by the end.

The images depict the stages of design analysis and refinement in a manufacturing context, focusing on the use of software and tools that aid in concept development, performance evaluation, optimization, and manufacturing assessment.

1. Concept Development & Iteration: This stage emphasizes the inception of a product where the primary objective is to transform ideas into tangible models. Iterative design and virtual prototyping software can play a pivotal role here, enabling rapid alterations and swift conceptual adjustments.

2. Performance Evaluation (Modeling & Simulation): At this juncture, the envisioned model undergoes rigorous virtual stress tests and simulations. Advanced software allows designers to anticipate how a product will perform under various conditions, effectively predicting its behavior and lifespan.

3. Model Optimization/Refinement: Optimization tools come into play to refine the model for peak performance, ensuring the product is not only functional but also economically and materially efficient. Here, the design is tweaked and adjusted to meet precise specifications and optimization goals.

4. Manufacturing Evaluation: In this final stage, the product design is evaluated for manufacturability. This encompasses assessing the feasibility of the production process, identifying potential issues, and ensuring the design is optimized for the chosen manufacturing methods, whether it's additive manufacturing, CNC machining, or injection molding.

These tools form an ecosystem that streamlines the design-to-production pipeline, ensuring a robust, efficient, and optimized manufacturing process, perfectly aligned with the capabilities and services offered by a vertically integrated company such as ADDMAN Group.

Prototyping Can be 3D printed at anytime during this process for evaluation.

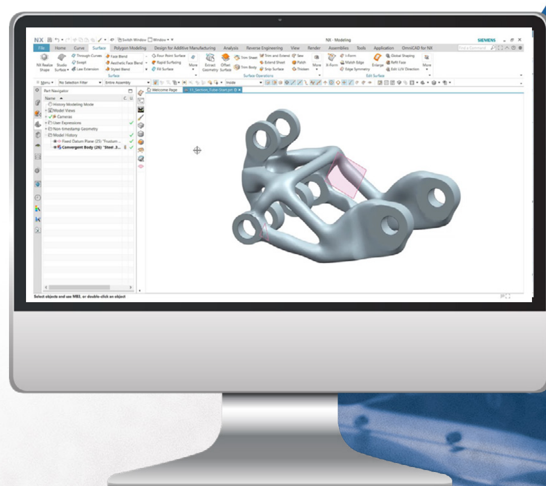
Concept Development & Iteration

The world of additive manufacturing is underpinned by powerful software and tools that assist designers in every stage of the design process. From concept development and iteration to performance evaluation and manufacturing optimization, these tools are essential for creating high-quality, functional parts.

In the initial stages of design, CAD (Computer-Aided Design) software plays a crucial role in developing and refining concepts. CAD allows designers to create detailed 3D models, including NURB (Non-Uniform Rational B-Splines) models and polygon meshes. NURB models are essential for representing complex curves and surfaces with high precision, while polygon meshes are used for creating more detailed and textured surfaces. These modeling techniques enable designers to visualize and iterate on their ideas rapidly, ensuring that the final design meets all functional and aesthetic requirements. For customers reading this eBook, understanding how CAD software facilitates quick iterations and accurate modeling is vital for appreciating the efficiency and precision it brings to the design process.

Computer-Aided Design (CAD)

- **A digital tool** used to design and draft both 2D drawings and 3D models.
- **Facilitates** the conceptualization, visualization, and modification or iterative designs.
- **Widely used in fields** like engineering, architecture, and product design.
- **Enables** accurate simulations, testing, and analysis of designs.
- **Serves as** data management and version control.
- **Some programs** allow you to build models with parametric features.



Autodesk:
- AutoCAD
- Fusion 360
- Inventor
- Revit

Dassault Systèmes:
- SOLIDWORKS
- CATIA
- DraftSight

PTC:
- Creo (formerly known as Pro/ENGINEER)

Siemens:
- NX (formerly known as Unigraphics)
- Solid Edge

Bentley Systems:
- MicroStation

Trimble:
- SketchUp

ANSYS:
- SpaceClaim

Hexagon:
- BricsCAD

Kubotek:
- KeyCreator

ZWSOFT:
- ZWCAD

CAD Design Software User Progression

Software	**Fusion360**	**Solidworks**	**Creo**
Relative Cost	$	$$	$$
Timeline to Proficiency	Weeks	Months	Year(s)

Concept Development & Iteration

3D Printing File Format Differences

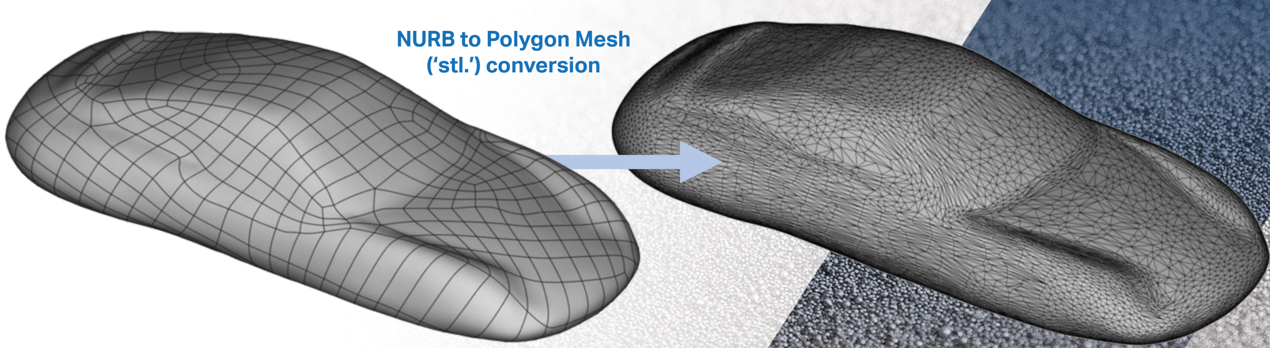
NURB models (“CAD files)

- **Definition:** A NURBS model consists of points connected by curves
- **Common NURB-based modeling programs:** Solidworks, Onshape, Pro-E, Creo
- **Typical users:** Engineers, CAD designers
- **Easier** to manipulate and edit
- **Can be converted** into multiple sub-CAD file types such as “STEP” or “PARASOLID” files

VS.

Polygon Meshes (“stl. files”)

- **Definition:** A polygon mesh consists of thousands or millions of small triangles
- **Common Polygon-based modeling programs:** Blender, Maya
- **Typical users:** 3D artists, Video/animation modeling
- **Commonly referred to as** stl. files
- **Used by 3D printers and software** to “slice” parts
- **Used as the generated file type** after 3D scanning
- **Difficult** to edit and convert back into other formats

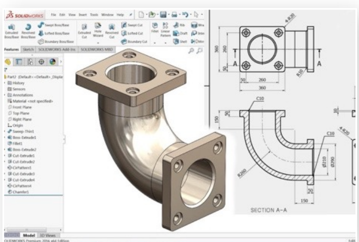


Concept Development & Iteration

Print-ready file requirements/guidelines

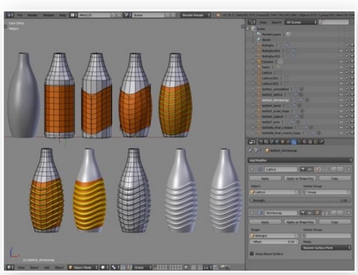
3D Scanning

- 3D Scanning produces "point-cloud" data
- Point cloud data must first be converted into an .stl format using software such as GeoMagics for the file to be ready for printing



3D CAD Design

- 3D CAD design produces a NURB based CAD file
- CAD files must also be converted into .stl



3D Poly Mesh Surface Modeling

- Poly Mesh produces a surface model
- Surface-based models must be converted to include volume-based data.
- Surface models cannot be used by the 3D printing software to slice.

Requirements:

- Model must be "watertight"
- All surface edges and bodies must be "touching" – no dis-connected models
- Model must at least be in a CAD or Poly based file format. (i.e. point cloud data unusable)

Point cloud data example

Example of solid versus 0 volume

Images via ikustec, Grabcad, pinterest

Performance Evaluation, Modeling, and Simulation

Once a concept is developed, performance evaluation through modeling and simulation tools becomes essential. These tools allow designers to test their designs under various conditions, predicting how they will perform in real-world applications. By simulating stress, thermal behavior, and material properties, designers can identify potential weaknesses and optimize their designs before manufacturing. This not only saves time and resources but also ensures that the final product meets all performance criteria. For customers, using these tools translates to more reliable and robust designs, reducing the risk of failure and improving overall product quality.

Performance Evaluation

Modeling & Simulation

Engineering simulation software has many uses, including volume and weight optimization, heat transfer analysis, stress and strain calculations, and fluid flow simulation.

This software removes the need for a physical prototype, making it possible to evaluate parts when they are very costly, dangerous, or difficult to test in real life.

Finite Element Analysis (FEA) software simulates and predicts the response of materials and structures to environmental factors, like force, heat, and vibration.

It's used to assess performance, identify potential failures, and optimize designs before physical prototyping.

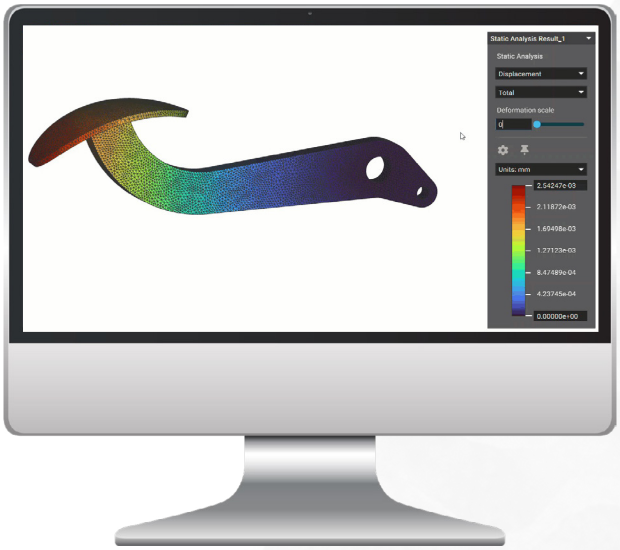


Image Source: nTop

Simulation software is integral to the additive manufacturing process, ensuring precision and reliability in part design and production. This software is a vital tool for engineers, allowing for complex calculations of volume, weight optimization, heat transfer, and fluid dynamics without the need for costly physical prototypes. Such capabilities are essential when working with high-risk or difficult-to-test components.

Model Optimization and Refinement

After performance evaluation, the next step is optimizing and refining the model for manufacturing. Predicting material behavior during the layer-by-layer deposition in additive manufacturing is challenging due to inherent variability in material properties, layer adhesion, and thermal behavior. Tools that simulate and compensate for potential distortions or imperfections are critical. These tools require a deep understanding of design parameters, material traits, and manufacturing conditions to ensure the final product maintains structural integrity, dimensional accuracy, and a high-quality surface finish. Simulation and compensation software streamline this process, making it easier for designers to produce parts that meet stringent quality standards. For readers, this means being equipped with the knowledge and tools to create superior products efficiently.

Topology Optimization Tools

These tools offer solutions to tackle the toughest engineering challenges, including lightweighting, thermal management, mass customization, architected materials, and manufacturing and tooling.

Latticing and lightweighting tools frequently form part of comprehensive software packages or are available as niche applications. They empower designers and engineers to craft and fine-tune lattice frameworks, often drawing upon robust FEA or other simulation data to deliver optimal outcomes.

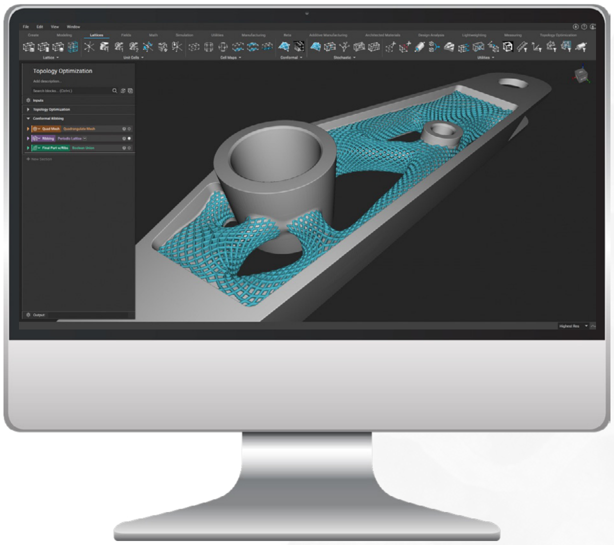


Image Source: nTop

In the realm of model optimization, the advent of topology optimization tools marked a significant shift towards designs that favor minimal material use. This focus on lightweighting gained momentum as additive manufacturing (AM) technologies emerged, enabling the creation of parts by adding material only where structurally necessary.

Manufacturing Evaluation

Simulation tools for the manufacturing process have become essential, akin to those used for assessing a model under loads. These tools have long been utilized in the machining industry through specialized programming software that generates visual renderings of machining programs to test for collisions or failures in system movements.

Simulation in Additive Manufacturing

In the additive manufacturing (AM) industry, simulation presents a unique challenge, particularly for metal processes. This complexity arises from numerous variables and interactions, including material properties, laser parameters, powder size distribution, and cooling rates. As a result, extensive research, especially at the university level, continues to address these challenges.

Advancements in Software Tools

Over the past few years, commercially available software tools for AM have significantly improved, thanks to substantial research from universities focused on metal additive manufacturing. These tools are now quite effective for most general applications, providing an early pass/fail assessment of part designs.

Polymer vs. Metal Processes

While these simulation tools are not as frequently used in the polymer side of the industry, due to fewer extreme variables during the process, they still hold value. For polymer users, these tools can verify how to adjust the model for different processes, ensuring accurate and efficient production.

Process Simulation & Compensation Tools

Predicting material behavior during layer-by-layer deposition is challenging due to inherent variability in material properties, layer adhesion, and thermal behavior.

Compensating for distortions or imperfections in AM demands a deep understanding of design parameters, material traits, and manufacturing conditions.

Simultaneously optimizing structural integrity, dimensional accuracy, and surface finish is streamlined with Sim & Comp software.

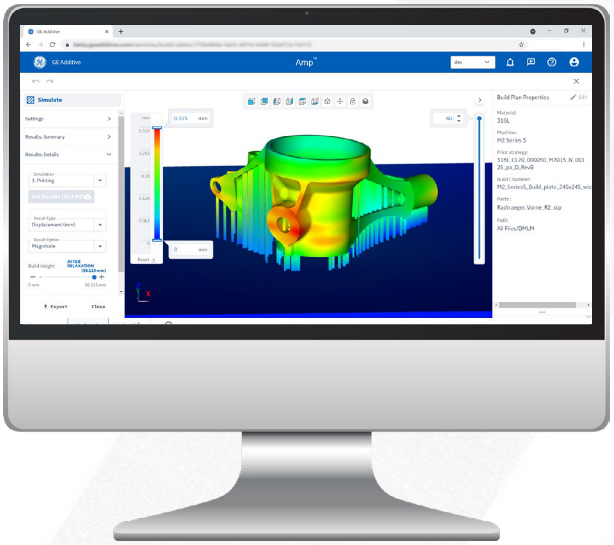


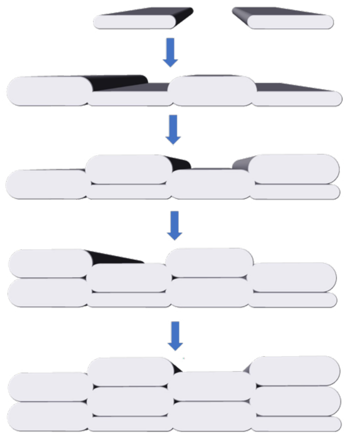
Image Source: Colibrium Additive

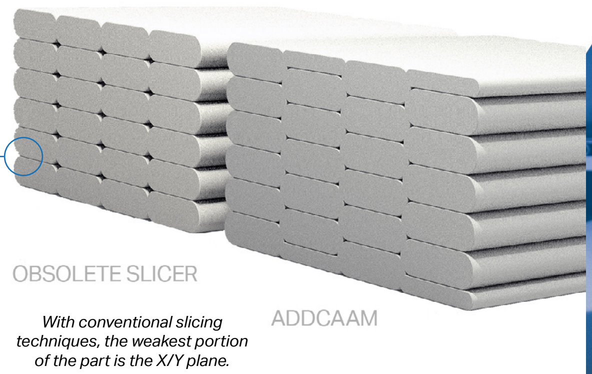
Innovation that Increases FDM Part Strength

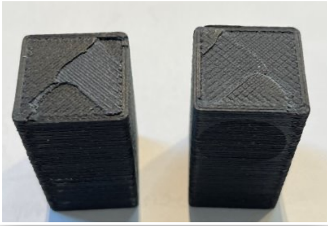
Traditional Fused Deposition Modeling (FDM) parts often suffer from weaknesses in the X/Y plane, leading to compromised strength and increased porosity.

These issues limit the performance and reliability of 3D-printed polymer components, particularly in demanding applications such as the aerospace and automotive industries. ADDMAN's ADDCAAM solution revolutionizes the 3D printing process by leveraging our proprietary CAAM (Computer Aided Additive Manufacturing) methodology. This cutting-edge software transforms conventional sliced files into an interlocking infill structure called InterFill 3D, producing parts that are 70% stronger and exhibit 100 times less porosity than industry standards. With ADDCAAM, manufacturers can achieve unparalleled strength and performance in their FDM parts, ensuring superior quality and reliability.

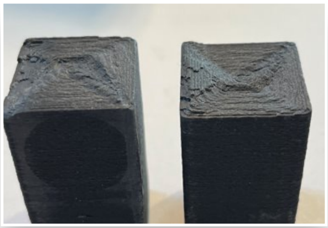
Offset Bead Print Sequence







Obsolete Slicer



ADDMAN CAAM

70%
STRONGER PARTS

Industry-leading part strength is enabled by cross-linking planes creating an innovative build technique.

100x
LESS POROSITY

The printing sequence of the offset beads allows for filling of the valleys, almost completely eliminating porosity.

What is CAAM?

CAAM or Computer Aided Additive Manufacturing is ADDMAN's approach to optimizing part strength. Our team challenges status-quo processes and develops new ways to advance software and material & machine parameters. Our goal is to make additive manufacturing a repeatable process, supporting both prototyping and mass production.

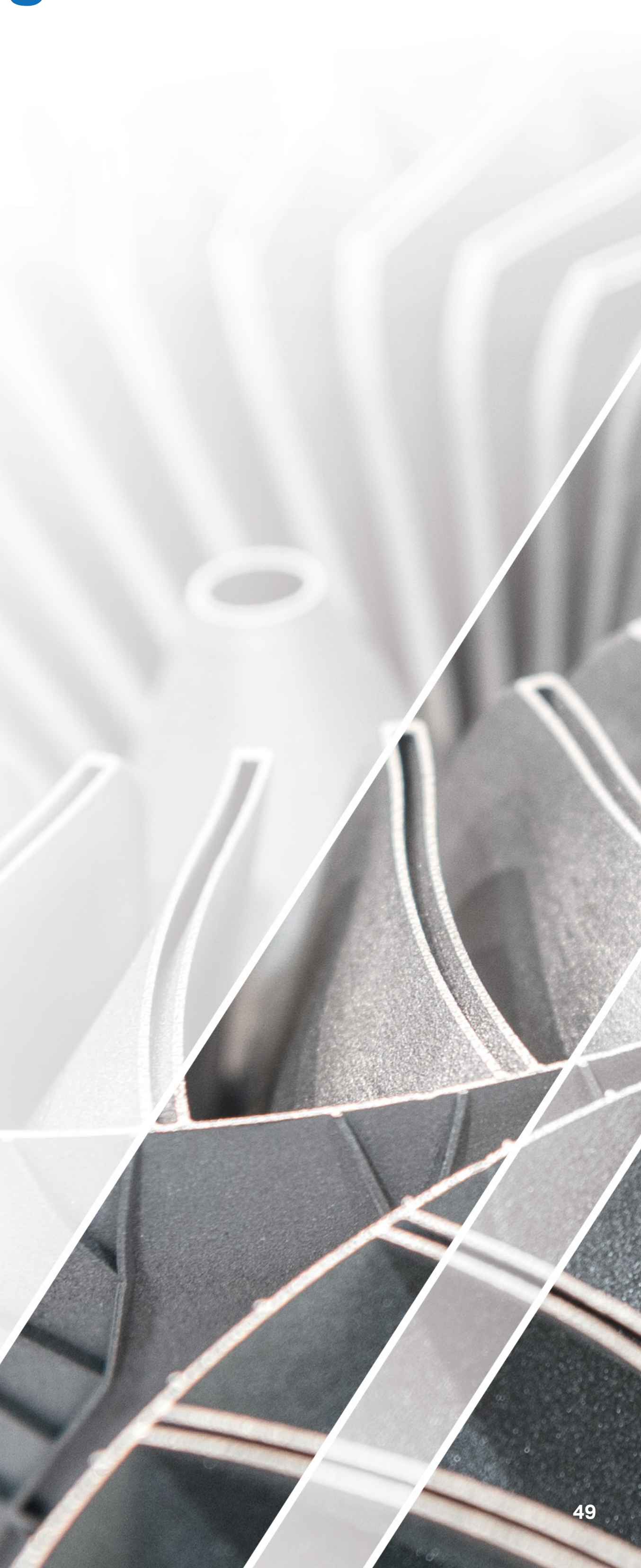
Click Here to see how ADDMAN is printing stronger parts

www.addmangroup.com/addcaam

Future Trends in Additive Manufacturing

The field of additive manufacturing is constantly evolving, with ongoing advancements in materials, processes, and technologies. Future trends include the development of multi-material printing, greater integration of AI and machine learning for optimized design and process control, and the expansion of AM applications in industries such as aerospace, healthcare, and automotive. As these trends unfold, staying informed and adaptable will be key to leveraging the full potential of additive manufacturing.

In summary, integrating DfAM principles and technical considerations into the design process is vital for maximizing the benefits of additive manufacturing. By understanding and applying these concepts, designers can create innovative, high-performance parts that push the boundaries of what is possible with traditional manufacturing methods.



Conclusion

As the industry continues to evolve, staying informed about the latest advancements and best practices is crucial. The knowledge shared in this eBook equips you with the foundational insights necessary to navigate the complexities of AM, from initial concept development to final production. By applying these principles, you can create innovative, high-quality parts that meet the demands of modern engineering and manufacturing. Embrace the opportunities offered by additive manufacturing, and take your designs to the next level, pushing the boundaries of what is possible in this exciting field.

ADDMAN®

CASTHEON
AN ADDMAN GROUP COMPANY

HARBEC
AN ADDMAN GROUP COMPANY

DINSMORE®
AN ADDMAN GROUP COMPANY

ADDMAN®
PRECISION