Carbon DLS™ 3D Printing Process Engineering Handbook

Table of Contents

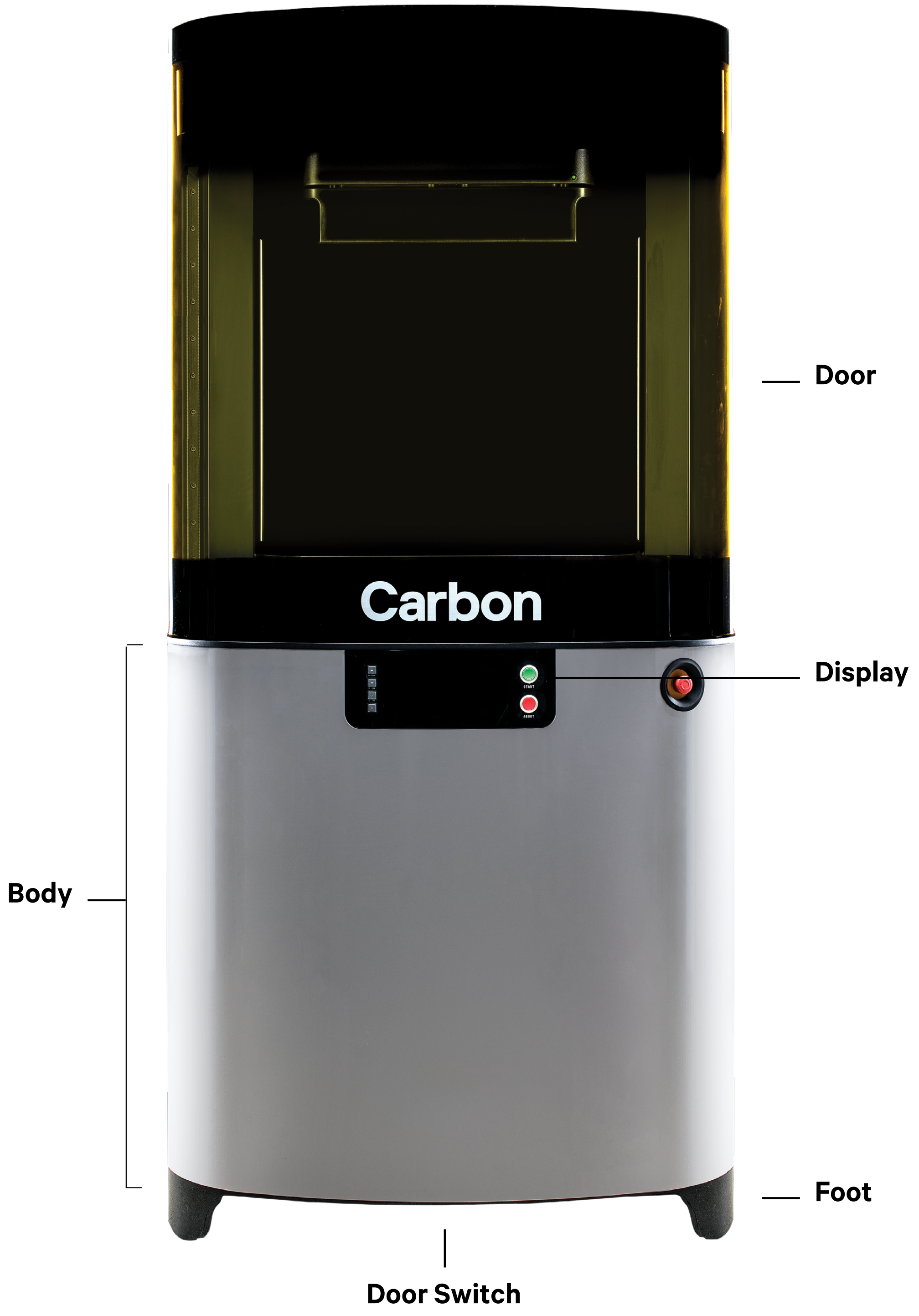
Overview

This handbook is your guide to the Carbon Digital Light Synthesis™ (Carbon DLS™) 3D printing process. Within these pages is a concise technical explanation of the technology, materials, and design best practices. The goal of this guide is to first help you decide if the Carbon DLS process is the right fit for your part, and to then achieve the best results no matter where you are in the product life cycle. By the end, you'll be able to design the best parts for your product without the design constraints associated with traditional manufacturing methods such as molding and machining.

If you are interested in experiencing parts printed with the Carbon DLS process, request sample parts by reaching out to us at sales@carbon3d.com.

Introduction

Carbon L1 Printer

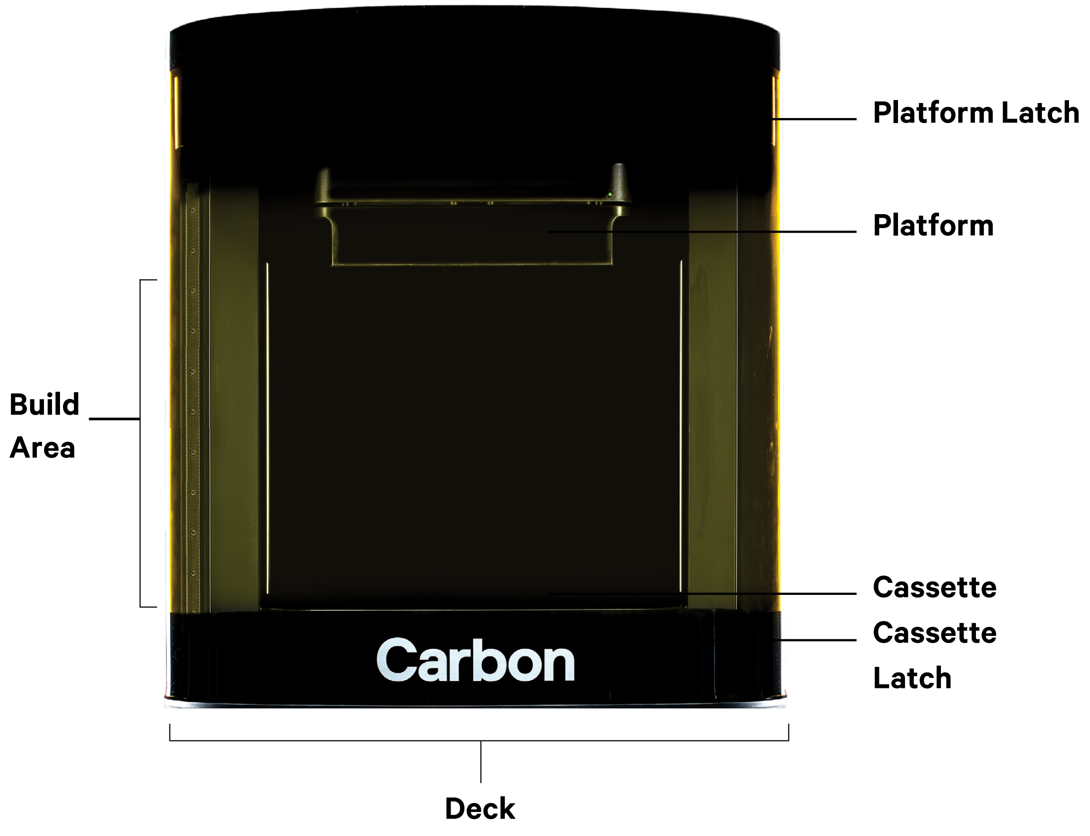


Carbon M2 Printer

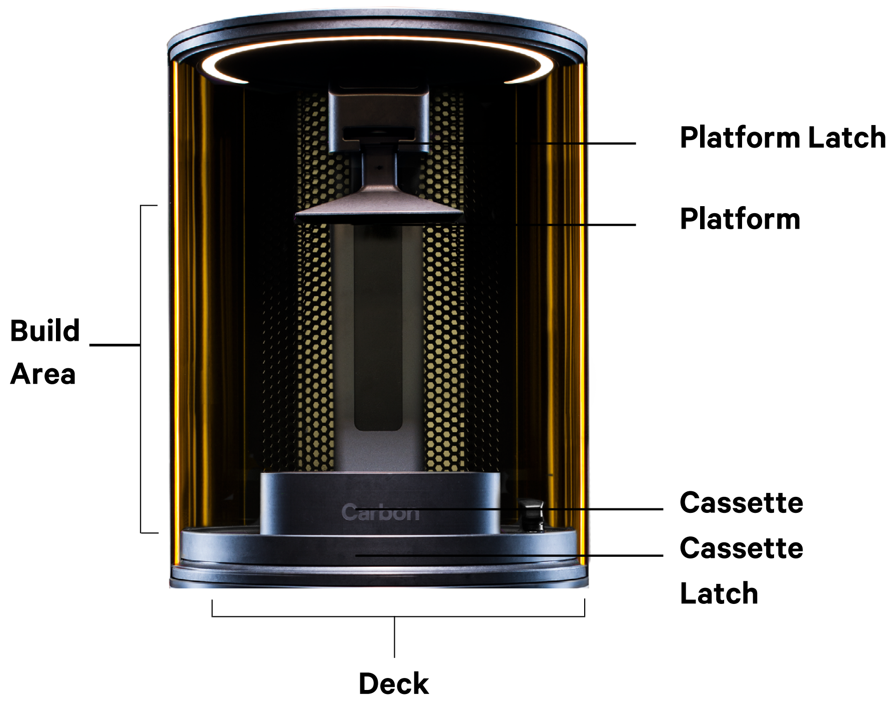
3D as It's Meant to Be.

We believe you should be able to access our 3D printing technology and expertise wherever, whenever, and however you need it so you can make better products, faster. From design to prototype to final product, each step toward creating a breakthrough product is vital. Both versatile and reliable, Carbon 3D printers allow you to deliver prototypes or finished goods, small batches, or large orders. Whether your goal is one part or one million custom pieces, we help you solve your product development challenges by empowering you with a deep knowledge of the Carbon DLS process and equipping you with a crucial set of best practices to use when designing parts for our platform. By understanding how to optimize your parts for throughput, accuracy, cost, or consistency, you can quickly get the most out of the Carbon DLS process and receive superior parts from Carbon production partners.

Carbon L1 Printer

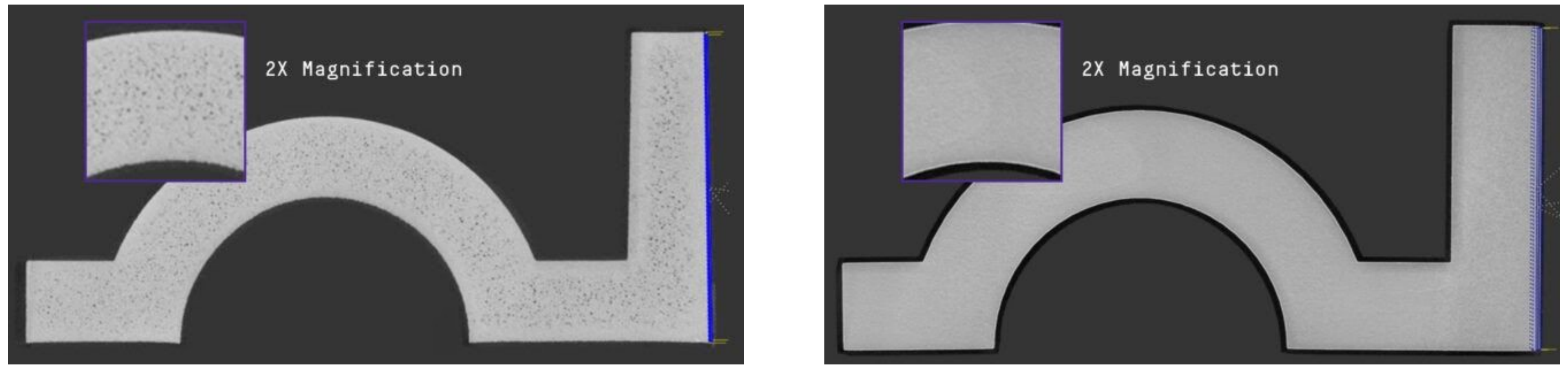


Carbon M2 Printer



1. Technology Background

	Carbon Digital Light Synthesis™ (Carbon DLS™)	Stereolithography (SLA)	Fused Deposition Modeling (FDM)	Digital Light Projection (DLP)	Powder Bed Fusion (SLS, MJF)
Wide range of real, functional parts	✓				
Excellent mechanical properties	✓				✓
Attractive surface finish	✓	✓		✓	
Fully dense	✓	✓		✓	



POWDER-BED FUSION

CARBON DLS™

Unlike powder-bed fusion 3D printing technologies, Carbon technology produces fully dense parts.

1.1 The Carbon DLS™ Process

Carbon DLS is a resin-based 3D printing process that utilizes digital light projection, oxygen permeable optics, and engineering-grade materials. This unique combination produces polymeric parts with exceptional mechanical properties, high resolution, and an attractive surface finish. The Carbon DLS™ process allows engineers and designers to iterate faster and deliver radically reimagined products.

Compared to other 3D printing technologies, the Carbon DLS process produces the widest range of real, functional parts. The surface finish, print speed, and material diversity options are better than powder-bed fusion processes, which are limited to PA 11, PA 12, or TPUs. The mechanical properties, print speed, and consistency are substantially better than traditional photopolymer technologies like SLA or DLP.

Because it can produce attractive, strong parts, the Carbon DLS process is used in a broad range of industries and across a part's life cycle, enabling teams to bring better products to market in less time. Thanks to the consistency of the Carbon platform, you can print parts reliably where you need them, both in the office and in manufacturing facilities, and produce the widest range of real, functional parts that are suitable for functional prototyping, low-volume production, and high-volume production. With greater design freedom, you can produce parts with geometries that are impossible for traditional manufacturing—like consolidated parts and dynamically adjustable lattice structures—which ultimately improve the parts' overall performance.

1.2 Fast Printing Via the “Dead Zone”

A key aspect of the Carbon DLS process is the “dead zone”—a thin, liquid interface of uncured resin between the window and the printing part. Oxygen presence allows light to pass through the dead zone, curing the resin above it to form a solid part without the part adhering to the window. Resin flows beneath the curing part as the print progresses, which maintains the “continuous liquid interface” that powers Carbon DLS and avoids the slow layering process inherent to many other resin-based printers.

1. Build Platform
2. Resin
3. Oxygen Permeable Window
4. Dead Zone
5. Light Engine

1.3 Mechanical Properties Set Through Heat

Traditional resin-based 3D printing processes produce weak, brittle parts. Carbon overcomes this by embedding a second heat-activated chemistry in our materials.

With our dual-cure resins, UV light sets the shape of a part as it prints on a Carbon printer. The part then undergoes a secondary heating step to set the properties. This secondary thermal cure forms a separate polymer network and results in robust, engineering-grade mechanical properties comparable to traditional manufacturing plastics, including injection molded parts.

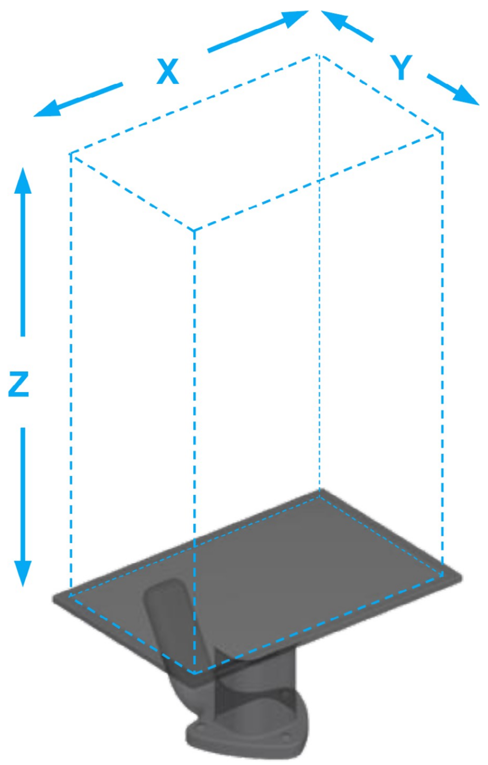
A part that is post-UV cure (left) and then post-secondary thermal cure (right), which sets the mechanical properties.

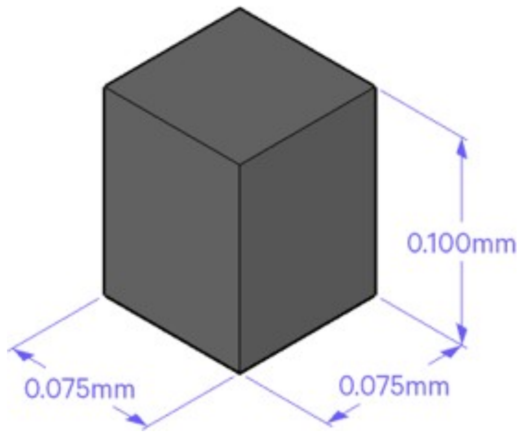
2. Carbon 3D Printers

Carbon 3D printers all utilize the Carbon DLS process. Therefore the design guidelines are approximately the same regardless of printer choice. The main difference between our printers involves the build area size, which determines both part size limits and resolution limits, and can make one more suitable than the other depending on the size and quantity of parts you desire. For example, for a part that just barely fits on the M2 printer vertically, the L1 printer may fit three or four parts easily, which can dramatically reduce print costs. These kinds of optimizations are highly dependent on the part but good to note.

2.1 M2 Printer

The M2 is our highest resolution, most accurate printer. It is ideal for smaller parts with intricate design features such as threading or internal channels.



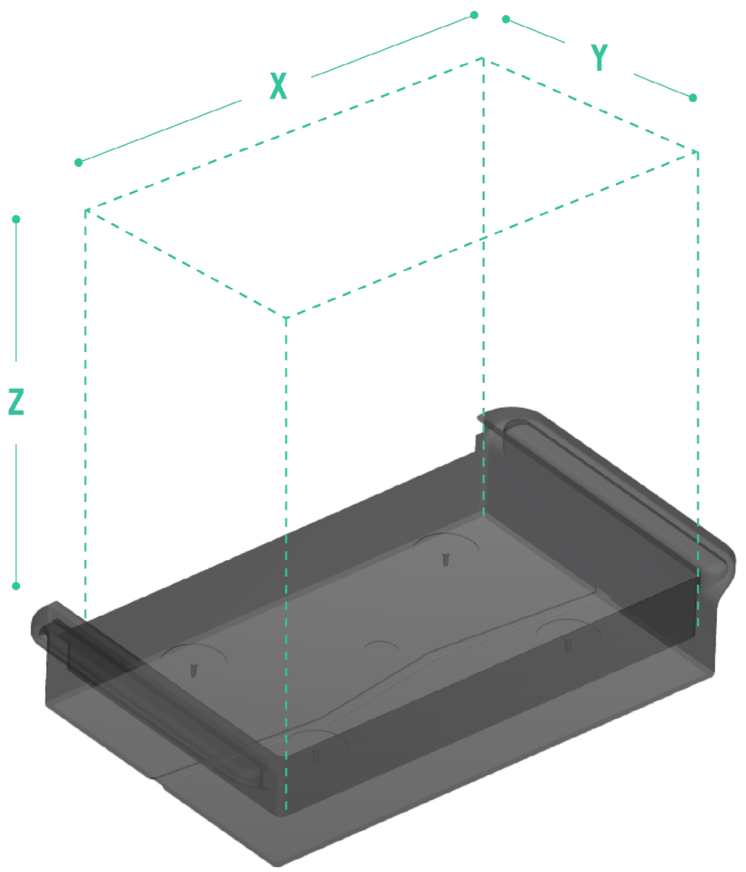


X	189 mm (7.4 in)
Y	118 mm (4.6 in)
Z	326 mm (12.8 in)

XY resolution	75 µm
Z resolution	100 µm, 50 µm, or 25 µm (adjustable)
Accuracy	±200 µm

2.2 L1 Printer

The L1 is our largest 3D printer and is best for consistent, high-volume production of large parts or many smaller parts. Companies like adidas, Specialized, and Riddell are using the L1 printer to scale production to millions of parts. Additionally, dental labs are utilizing the L1 printer to quickly ramp up clear aligner model production.



X	400 mm (15.7 in)
Y	250 mm (9.8 in)
Z	508 mm (20 in)

XY resolution	160 µm
Z resolution	100 µm, 50 µm, or 25 µm (adjustable)
Accuracy	±300 µm

2.3 Smart Part Washer

The Smart Part Washer optimizes post-processing for your part. Our software provides automatic, specific wash protocols that allow you to create a repeatable washing process so you can reduce manual labor and improve part quality.

3. Carbon Materials Overview

Carbon offers a wide range of best-in-class 3D printing materials used to create isotropic parts with exceptional surface quality, precise features, and optimal material performance. With Carbon materials and the Carbon DLS process, you can produce engineering-grade parts with predictable mechanical responses, enabling you to seamlessly transition from design to functional prototyping and end-use production, all on the same platform.

Carbon dual-cure and single-cure resins are developed in-house and are produced by leading chemical suppliers like DuPont and EssTech. Carbon has also partnered with leading suppliers to offer premium third-party materials.

3.1. Rigid Materials

Loctite IND405 Clear

FPU 50

UMA 90

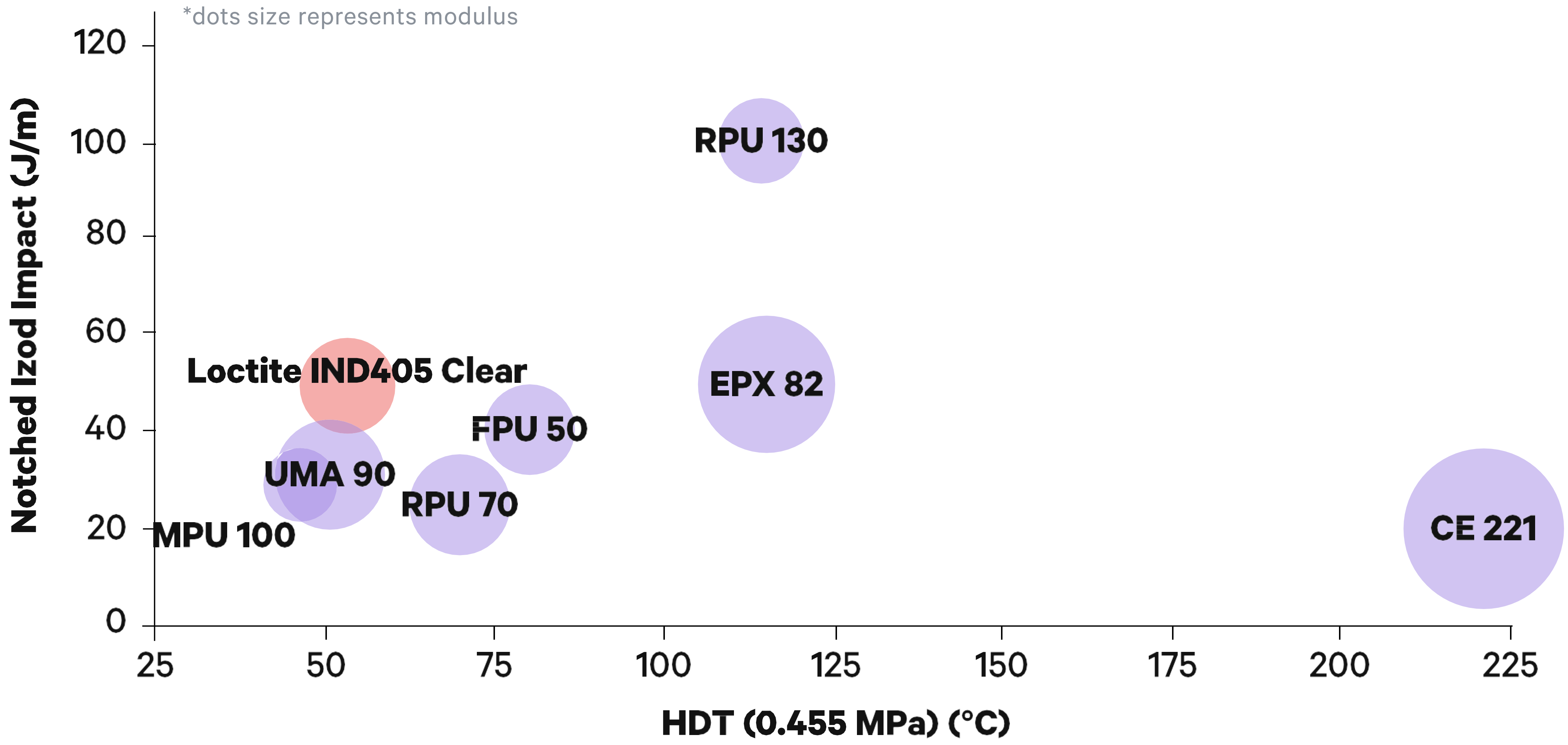
MPU 100

EPX 82

RPU 70

RPU 130

CE 221

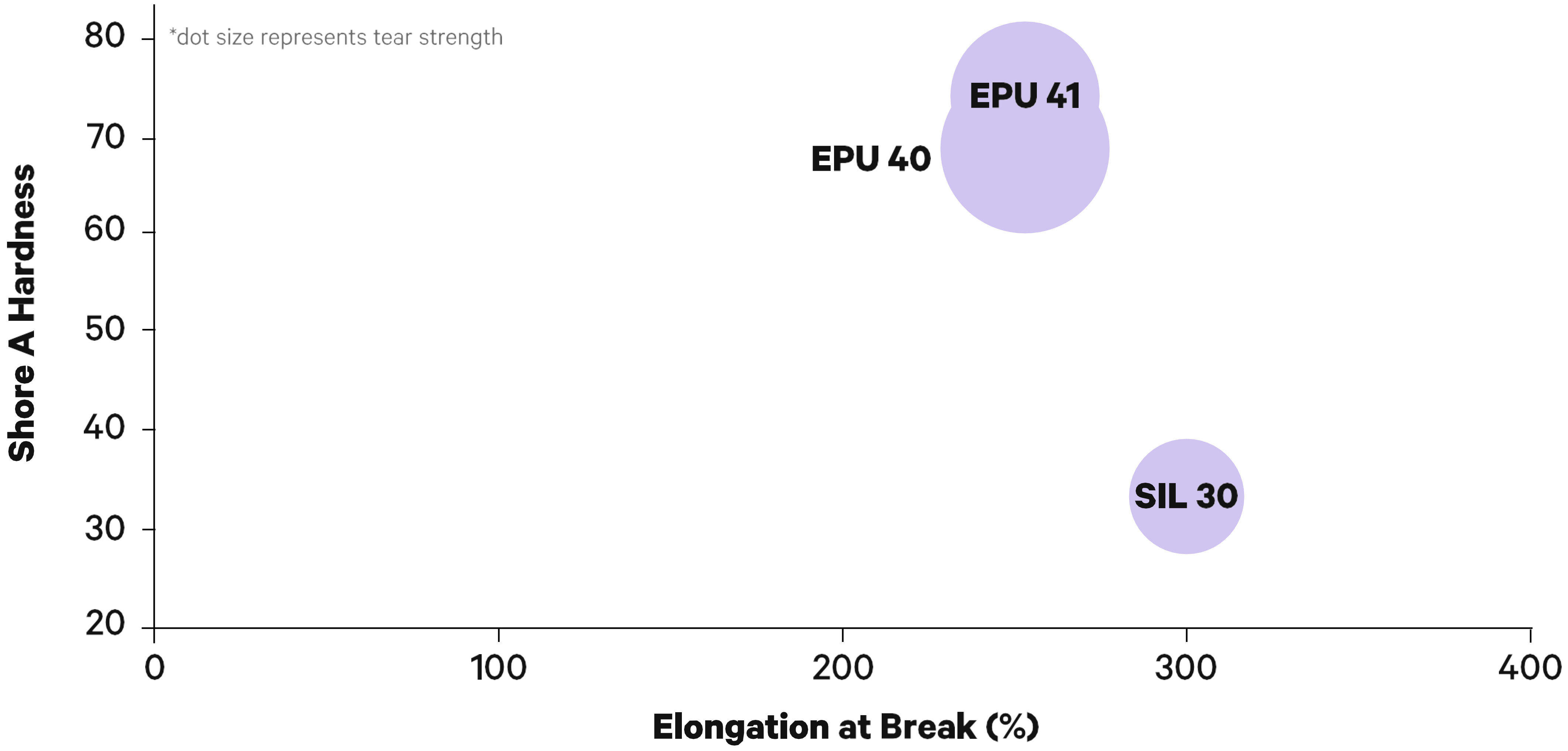


3.2 Elastomeric Materials

EPU 40

EPU 41

SIL 30



3.3 Carbon Material Properties

What material performance is required for your part(s)? Are there specific tolerances or mechanical properties that need to be met? Are there specific environmental conditions the part will need to withstand? Carbon material properties can be leveraged to meet your specific part requirements.

	RESIN	ULTIMATE TENSILE STRENGTH	ELONGATION AT BREAK	TENSILE MODULUS	SHORE HARDNESS	IMPACT STRENGTH *	HEAT DEFLECTION TEMP**
2 PART	CE 221	85 MPa	3%	3900 MPa	92D	15 J/m	230° C
	EPX 82	80 MPa	5%	2800 MPa	89D	45 J/m	130° C
	RPU 130	35 MPa	100%	900 MPa	77D	75 J/m	120°C
	RPU 70	40 MPa	100%	1700 MPa	80D	15 J/m	60° C
	FPU 50	25 MPa	200%	700 MPa	71D	40 J/m	70° C
	MPU 100	35 MPa	15%	1300 MPa	81D	30 J/m	50° C
	EPU40	9 MPa	300%	N/A	68A	N/A	N/A
	EPU 41	15 MPa	250%	N/A	73A	N/A	N/A
	SIL 30	3.5 MPa	350%	N/A	35A	N/A	N/A
1 PART	HENKEL IND405	42 ± 4 MPa	120 ± 8%	1500 ± 31 MPA	78D	50 J/m	53° C
	DPR 10	45 MPa	>4%	1800 MPa	N/A	20 J/m	61° C
	UMA 90	30 MPa	30%	1400 MPa	86D	30 J/m	45° C

▇ Indicates the highest value in its category.

Note: Henkel is responsible for LOCTITE® 3D IND405 Clear material testing, evaluation, and reporting. You can find more detail on its material properties here.

3.4 Comparable Thermoplastics

Was the part made in a comparable material previously? Do internal or industry standards specify a particular material? Carbon materials are comparable to the common thermoplastics listed in this chart:

Resin	Comparable Thermoplastic
CE 221	Glass-filled nylon
EPX 82	20% glass-filled PBT
RPU 130	Nylon 6
RPU 70	ABS or PC ABS
FPU 50	Polypropylene
MPU 100	Medical-grade ABS
EPU 40	TPU
EPU 41	TPU
SIL 30	TPE
IND405	Polypropylene

3.5 Chemical Compatibility and Sterilizability

Will your part(s) be exposed to chemicals? Determine which Carbon material will meet your chemical compatibility and/or sterilizability testing by factoring in the length of chemical exposure and utilizing the following chart.

Currently, Carbon has one "standard" chemical compatibility test for all our materials. We evaluate our materials per ASTM D4060 on the basis of percent weight gain of a fully submerged standard sample after one week. This data tells you exactly how much weight the sample gained after being fully submerged in the solution at room temperature and normal atmospheric pressure. It is assumed that weight gain over 5% starts to indicate material degradation and incompatibility.

This test is aggressive in the sense that it requires a full and continuous submersion. It is important to take into consideration the functional application of a product in service when considering this data.

Unless otherwise noted, this weight gain does not necessarily specify how the material degrades, if at all. While high weight gain does commonly accompany dimensional changes, the change doesn't reflect in what manner, and there is no established relationship. In addition, the weight gain also does not note color change or effect on material properties, such as tensile performance, impact strength, etc. Color change is not reported in the official release. Reach out to our materials team to see if this information is available for a specific material and solvent.

Chemical Compatibility

Note: Due to variability in part geometry and level of exposure in actual use, it is required that adequate validation is done for production applications.

		Mass Gain* (%)						
CLASS	CHEMICAL	Rigid Resins				Elastomeric		
		CE 221	EPX 82	RPU 70	RPU 130	EPU 40	EPU 41	SIL 30
Household Chemicals	Bleach (NaClO, 5%)	<5%	<5%	<5%	-	<5%	<5%	<5%
	Sanitizer (NH4Cl, 10%)	<5%	<5%	<5%	-	<5%	<5%	5 – 15%
	Distilled Water	<5%	<5%	<5%	-	<5%	<5%	5 -15%
	Sunscreen (Banana Boat, SPF 50)	<5%	<5%	<5%	5 -15%	5 -15%	>30%	5 -15%
	Detergent (Tide, Original)	<5%	<5%	<5%	-	<5%	5 -15%	5 -15%
	Windex Powerized Formula	<5%	<5%	<5%	-	5 -15%	5 -15%	5 -15%
	Hydrogen Peroxide (H2O2, 30%)	<5%	<5%	<5%	-	15 -30%	15 -30%	15 -30%
	Ethanol (EtOH, 95%)	<5%	5 -15%	15 -30%	-	>30%	>30%	>30%
Industrial Fluids	Engine Oil (Havoline SAE 5W-30)	<5%	<5%	<5%	<5%	<5%	<5%	<5%
	Brake Fluid (Castrol DOT-4)	<5%	<5%	<5%	-	15 -30%	15 -30%	🟧
	Airplane Deicing Fluid (Type I Ethylene Glycol)	<5%	-	<5%	-	<5%	-	<5%
	Airplane Deicing Fluid (Type I Propylene Glycol)	<5%	-	<5%	-	<5%	-	5 -15%
	Airplane Deicing Fluid (Type IV Ethylene Glycol)	<5%	-	<5%	-	<5%	-	<5%
	Airplane Deicing Fluid (Type IV Propylene Glycol)	<5%	-	<5%	-	<5%	-	5 -15%
	Transmission Fluid (Havoline Synthetic ATF)	<5%	<5%	<5%	<5%	<5%	<5%	<5%
	Engine Coolant (Havoline XLC, 50%/50% premixed)	<5%	<5%	<5%	-	<5%	-	<5%
	Diesel (Chevron #2)	<5%	<5%	<5%	15 -30%	>30%	>30%	15 -30%
	Gasoline (Chevron #91)	<5%	-	>30%	-	>30%	-	>30%
	Skydrol 500B-4	<5%	<5%	5 -15%	-	>30%	>30%	>30%
Strong Acid/Alcohol/Base	Sulfuric Acid (H2SO4, 30%)	<5%	<5%	<5%	-	>30%	15 -30%	>30%
	Sodium Hydroxide (NaOH, 10%)	<5%	<5%	<5%	-	<5%	-	<5%

*Percentages are percent weight gain after a 1 week submersion per ASTM D543. Values do not represent changes in dimension or mechanical properties.

Note: DPR 10, PR 25, and UMA are not included in chemical testing because they are used primarily for prototyping and models. MPU 100 is not included because it is primarily used for medical applications and would not be tested in industrial fluids.

3.6 Biocompatibility Testing

All of our materials pass basic biocompatibility testing (cytotoxicity, irritation, sensitization) unless otherwise indicated. For detailed testing results of a specific material, go to the biocompatibility section of the material's TDS listed in the following chart.

	Resin	Specific Testing Results via TDS*
2-Part	CE 221	TDS: CE 221
	EPX 82	TDS: EPX 82
	RPU 130	TDS: RPU 130
	RPU 70	TDS: RPU 70
	FPU 50	TDS: FPU 50
	MPU 100	TDS: MPU 100
	EPU 40	TDS: EPU 40
	EPU 41	TDS: EPU 41
	SIL 30	TDS: SIL 30

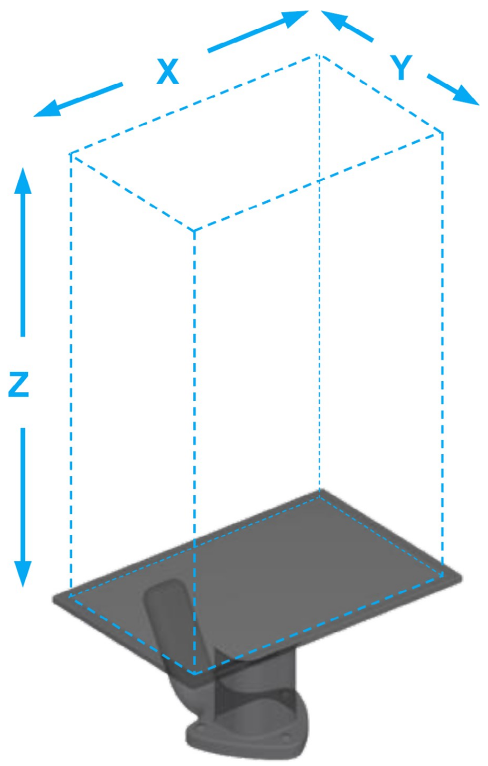
*Disclaimer: Biocompatibility results may vary based on printing and/or post-processing procedures.
Note: DPR 10, PR 25, and UMA pass basic biocompatibility testing but are not included in the chart because they are used primarily for prototyping and models.

For example, MPU 100, a material primarily used in medical applications, passed USP-VI testing and proved to be compatible with a range of commonly used hospital disinfectant agents (ethanol, bleach, chlorhexidine gluconate (CG), benzalkonium chloride (BC)) after showing minimal changes to the tensile properties as well as no change in mass, dimensions, or color.

4. Printing Overview

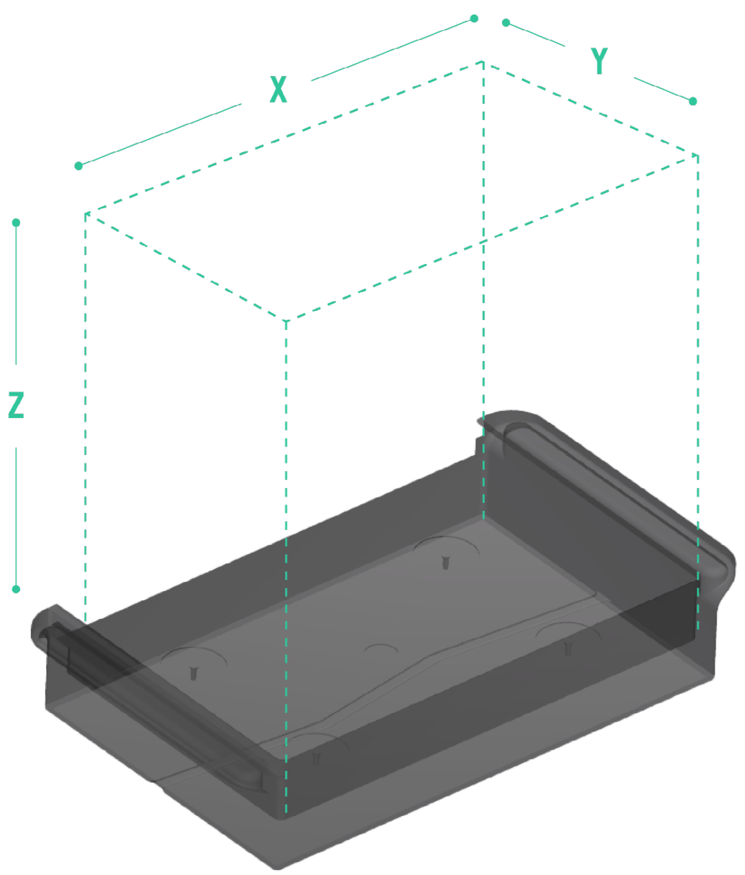
4.1 Build Envelope

Our process can accommodate single or multiple parts provided that they fit within the build envelope of your specific printer.



Carbon M2 Printer

X 189 mm (7.4 in)
Y 118 mm (4.6 in)
Z 326 mm (12.8 in)



Carbon L1 Printer

X 400 mm (15.7 in)
Y 250 mm (9.8 in)
Z 508 mm (20 in)

4.2 Advantages of the Carbon DLS™ Process

4.2.A A Flexible Manufacturing Solution

The Carbon DLS process can transition from prototyping to production needs, multiplying the value of Carbon printers as a piece of manufacturing equipment. Below are customer examples that demonstrate the benefits of Carbon's flexibility, including design freedom and economic advantages.

4.2.B Economic Advantages

Low Volume:
Cost effectively produce low volumes (under 10,000 units)

Ford Super Duty Truck:
Aptiv trailer tow connector cap

On Demand:
Eliminate warehousing inventory needs

Ford Focus: HVAC lever arm service parts

Quick Turnaround:
Meet short deadlines

NASA Seeker robot also intricate channeling

Rapid Design Iterations + Functional Prototyping:
Reduce product development time and shorten time-to-market

Lamborghini air vents

Cost-Effective Customization:
Customize products for a more customer-centric experience

MyFit Solutions x ErPro earbuds

4.2.C Design Freedom

Unmoldable Geometries

Create radical new designs with geometries that would be impossible to mold or machine


adidas 4D midsole


Riddell helmet


Specialized saddle

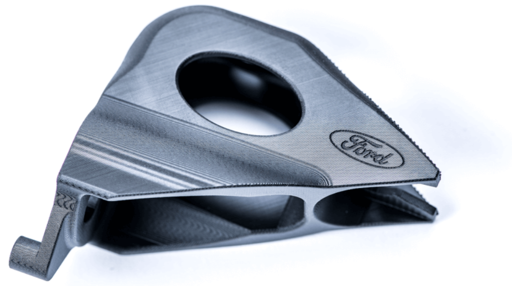
Lattices

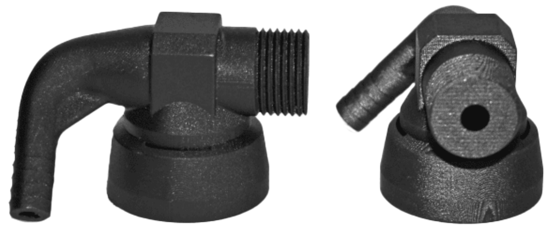
Foam replacement (cushioning, vibration isolation, impact absorption, energy return) for the adidas 4D midsole, Riddell helmet, and Specialized saddle

Part Consolidation

Minimize assembly costs


Vitamix (3 to 1)


Ford Mustang GT 500: Electric parking brake (EPB) bracket (3 to 1)


Paragon x Agrifac (6 to 1)

Single-Process Surface Finishes

Reduce post-processing costs


Ford Raptor: Low-volume cosmetic plugs for niche markets


Lamborghini Urus Super SUV: Textured fuel cap

Other common production efforts

- Housings
- Electrical connectors
- Foam replacement
- Skin-contact applications
- Single-use surgical instruments
- Wearable technology
- Complex water and air handling
- Fixtures for baking
- And more!

4.3 Accuracy

4.3.A Default Accuracy

The Carbon DLS process comfortably delivers parts at ±200 µm and often ±100 µm when parts meet recommended feature sizes (see section 5.2 in design guidelines).

4.3.B Tuning Tighter Than Default Accuracy

Similar to traditional production techniques, the Carbon DLS process is tunable, so you can achieve tighter tolerances than default. Through a combination of part-file optimizations, print optimizations, and iteration, you can expect to achieve critical dimensions as tight as ±50 µm. This often takes considerable engineering effort and time.

The Carbon DLS process is a highly controllable process, though it has many possible points of variation. Tuning is the process of identifying and stabilizing these variables. Any variation in accuracy or tolerances is dependent on a resin’s specific properties, print speed, the quality of projected image relative to the cleanliness of the light path (deck window, cassette window), and mechanical tolerances of the various components of each individual printer. Because tolerances are tighter in the print plane (XY), features that require a higher degree of accuracy should be parallel to it. Additionally, we are continually working to characterize and improve upon the accuracy of our engineering resins.

4.4 Printing Resolution

Conceptually, printing a part is effectively playing a movie of your part. An STL file is converted into slices, with each slice containing the 2D shape of what should be printed at that step in the vertical Z plane. The UV light engine plays the movie as it projects slice “frames” into the liquid resin. The first slice cures the part to attach to the build platform. Then the following slices shape the part as the platform continues to rise out of the resin.

Therefore each printed part is made of consecutively printed slices. A slice is a field of pixels that are projected by the light engine and cure the liquid resin. Each pixel is a 75 µm (0.075-mm/0.003-in) square. The height of the pixels is determined by the slice thickness, which is typically 100 µm (0.100 mm/0.004 in), but can be adjusted to 50 µm or 25 µm when parts require a higher level of feature detail.

Note that the thinner the slice, the longer the overall print time. When slices are very thin, more UV light projections have to take place to form the entirety of your part. This generates more heat, which causes the print to slow down so the printer can keep operating temperatures within acceptable limits (see section 6.3.A for more detail on heat generation as a force that affects printing).

Print process

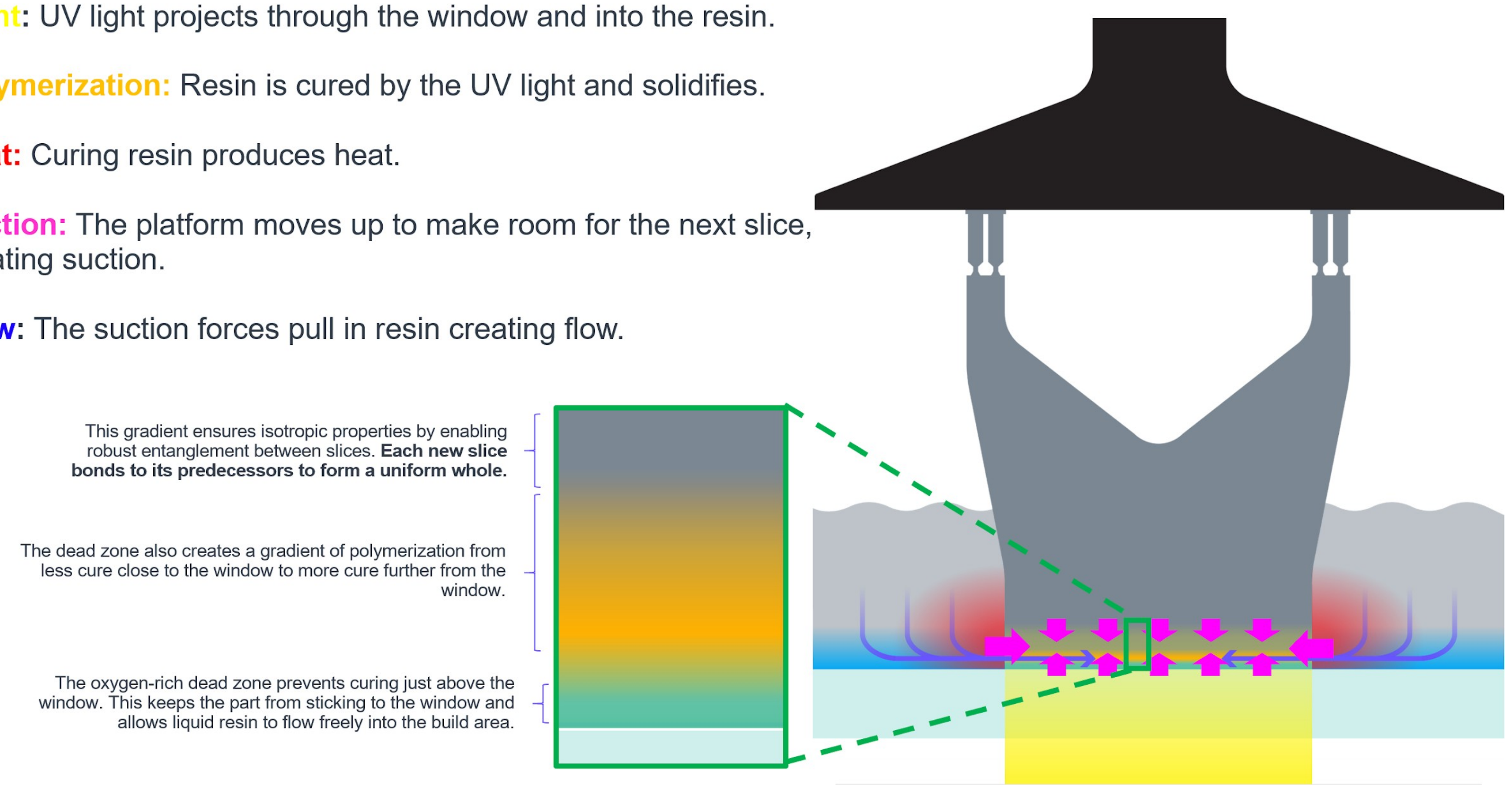
Light: UV light projects through the window and into the resin.

Polymerization: Resin is cured by the UV light and solidifies.

Heat: Curing resin produces heat.

Suction: The platform moves up to make room for the next slice, creating suction.

Flow: The suction forces pull in resin creating flow.



This gradient ensures isotropic properties by enabling robust entanglement between slices. **Each new slice bonds to its predecessors to form a uniform whole.**

The dead zone also creates a gradient of polymerization from less cure close to the window to more cure further from the window.

The oxygen-rich dead zone prevents curing just above the window. This keeps the part from sticking to the window and allows liquid resin to flow freely into the build area.

4.4.A Isotropy with Slices

In other vat photopolymerization technologies, the final parts often have “layers”, with each layer cured independently of the next. This creates many issues for parts; shale-like layers may not adhere fully to one another, resulting in significant mechanical differences in different orientations. Further, if a gap is large enough for air to penetrate the part, you may experience problems including diminished heat resistance and increased fracture modes.

In the Carbon DLS 3D printing process, the slices are continuously cured to prevent layer adhesion, and the secondary thermal cure crosslinks polymers across slice boundaries, creating fully isotropic parts.

4.5 Optical Effects

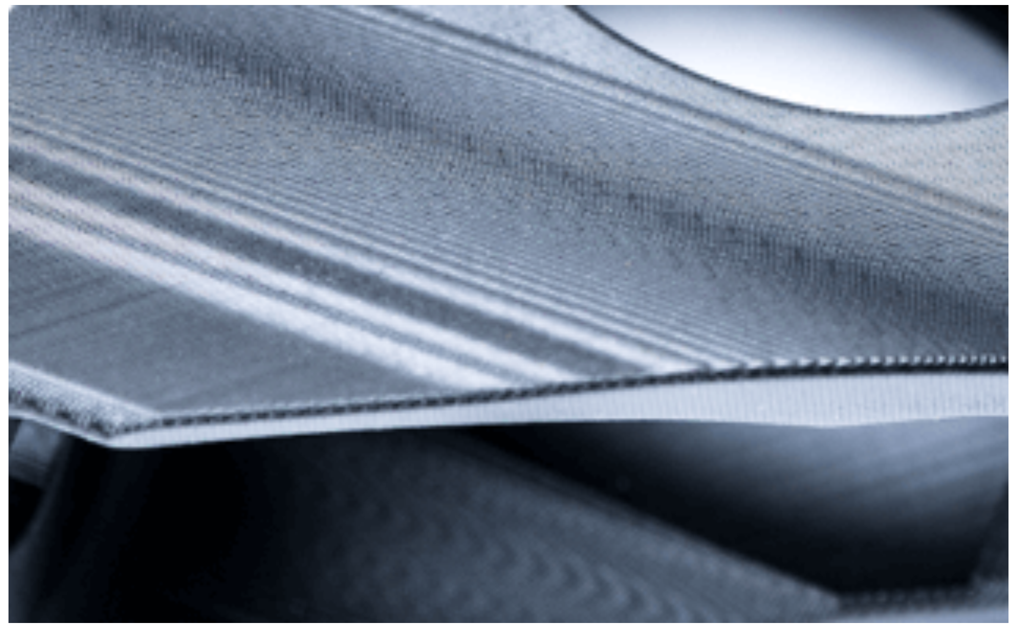
In this section we cover the main optical effects you might encounter while utilizing the Carbon DLS™ process. In general, to avoid optical effects, we suggest designing parts with larger feature sizes when possible. If your part design requires small* features, you should expect to have to iterate on the design to offset potential optical effects.

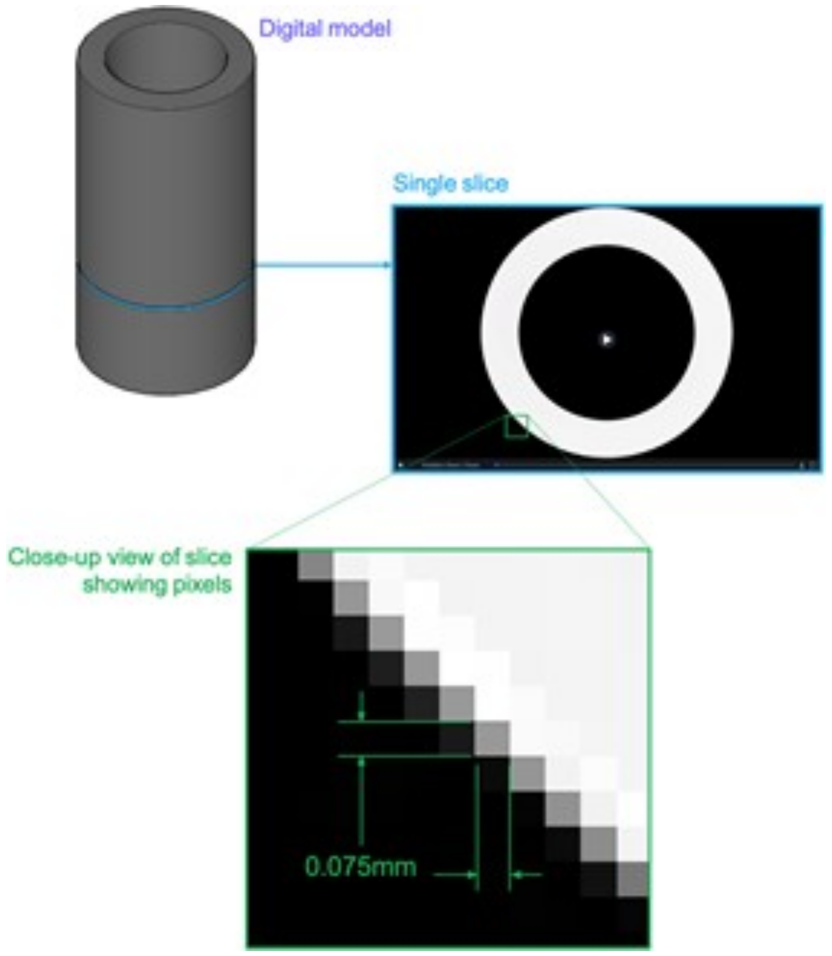
*Carbon defines “small” as: a part that fits in a 10-mm cube, holes that are <2 mm, cavities that are <4 mm.

4.5.A Moire Pattern

The following part showcases an optical effect known as the Moire pattern that is common in UV photopolymer printing processes. The usual misconception is that the pattern lines are layers. In actuality, you are observing pixelation at the part’s edges, where light is distributed and scattered. These pixel artifacts do not affect the part’s isotropic nature or mechanical performance.

If aesthetics are an important feature of your part, and you would prefer a different surface finish, we offer tools to help you. You can apply a surface texture through our software texturing tool for a matte finish, a functional texture to improve grip, and more. You can also utilize common finishing techniques through post-processing to achieve a gloss finish. See more on optimizing for surface finish and aesthetics in section 6.4.C.



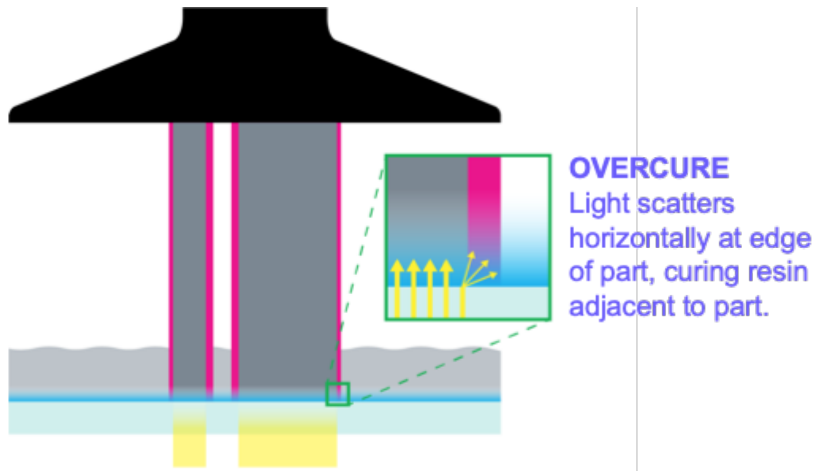


4.5.B UV Curing Abnormalities

Because resin is not totally opaque to UV light, the light can penetrate through thin areas of material and cause inaccuracies known as overcure and cure-thru. This is not noticeable in most resins unless you are trying to reach very tight tolerances, print very small parts, or the part is under the recommended minimum feature sizes (see section 5.2 design guidelines). Materials that are white or clear exhibit these optical effects the most.

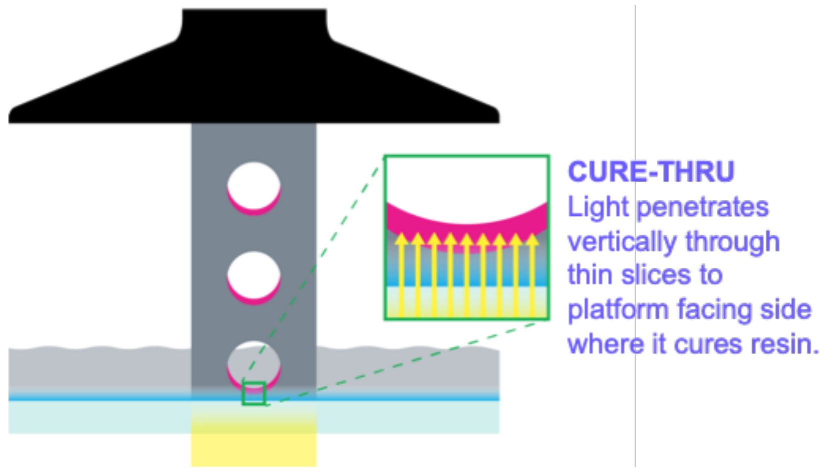
4.5.B.A XY plane: Overcure

Overcure is an effect caused by light scattering horizontally at the edges of a slice, where the material is thinner and less opaque. This scattered light cures resin adjacent to the part and typically causes an additional 0.010–0.075 mm of part curing. This effect is greater in smaller cavities due to higher local temperatures.



4.5.B.B Z axis: Cure-thru

Cure-thru occurs when light penetrates vertically through a thin slice of material and cures resin on the back side of the slice. It is essentially overcure in the Z axis. This phenomenon causes holes to be oblong and 0.050–0.200 mm smaller than nominal in the Z axis.



5. Design Guidelines for Carbon DLS™ Process

Once you have determined that your part is a good fit for the Carbon DLS™ process, the next step is to review your part design and features through the lens of our Carbon DLS design guidelines. Similar to injection molding, this is the step when you would evaluate the features of your part against that manufacturing process' specific set of design guidelines to ensure its manufacturability. However, unlike injection molding, the speed of the Carbon DLS process helps you iterate significantly faster and cheaper than you could with traditional tooling costs and timelines.

It's important to follow these guidelines when designing and preparing your part so you can avoid print failure, reduce potential defects, eliminate human error, and reduce post-processing time. Below is a short summary of design principles and tips we will dive deeper into in the following sections.

5.1 Main Design Principles

- Gradual geometry changes
- Consistent wall thickness
- No trapped volumes
- Self-supported parts
- Cleanable part design

5.1.A Supporting Design Tips

- Reduce sharp edges or corners
- Bring the part to meet the build platform
- Tie thin/delicate features into your part for stability in bake
- Use organic/natural designs when possible
- Use the maximum overhang or bridge distance for feature spacing
- Threaded holes and metal inserts are usable with Carbon

5.2 Begin with Recommended Feature Sizes

Are your features properly sized for successful printing? Refer to the following chart as a starting point when designing your part's feature sizes so you can maximize the printability of your part. Note that you can adjust these feature sizes through further print optimizations later on (see section 6 optimizing). If your part's feature sizes are drastically outside the following recommended ranges, the Carbon DLS process is probably not a good fit for producing your part.

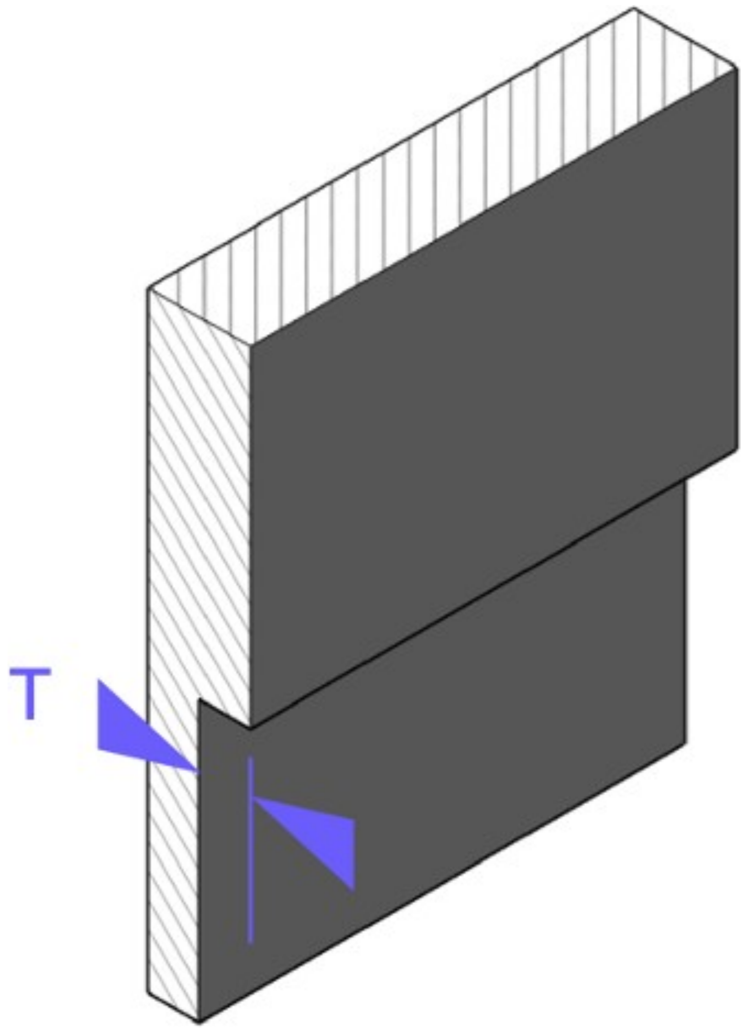
Feature	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Wall Thickness Unsupported Recommended Minimum (mm)	2.5	2.5	2.5	2.5	2.5	2.5	2.5	2.5	2.5	2.5
Wall Thickness Supported Recommended Minimum (mm)	1.0	1.5	1.0	1.0	1.0	1.0	1.5	1.0	1.0	1.5
Overhangs Recommended Maximum (mm)	2.0	2.0	2.0	1.0	2.0	3.0	2.0	3.0	3.0	1.0
Bridges Recommended Maximum (mm)	4.0	4.0	6.0	2.0	4.0	6.0	4.0	6.0	6.0	2.0
Unsupported Angle Horizontal (degree)	30	40	40	40	35	40	40	30	30	40
Positive Features Horizontal XY Recommended Maximum (mm)	0.4	0.3	0.4	0.5	0.5	0.4	0.3	0.6	0.4	1.0
Positive Features Vertical Z Recommended Maximum (mm)	0.2	0.3	0.2	0.2	0.2	0.2	0.2	0.2	0.2	1.0
Holes Horizontal XY Recommended Minimum (mm)	0.5	0.5	0.9	0.5	0.5	1.0	0.6	0.9	0.9	2.0
Holes Vertical Z Recommended Minimum (mm)	0.6	0.5	0.8	0.5	0.5	0.7	0.9	0.6	0.8	2.0
Clearance for Mating Parts Recommended Minimum (mm)	0.4	0.5	0.5	0.5	0.5	0.8	0.4	0.5	0.5	0.5
Engraving Depth/Embossing Height Recommended Minimum (mm)	0.3	0.3	0.3	0.3	0.3	0.4	0.3	0.3	0.3	0.5
Engraving and Embossing Text Size Recommended Minimum (mm)	3.0	3.0	3.0	3.0	3.0	3.0	3.0	3.0	3.0	3.5

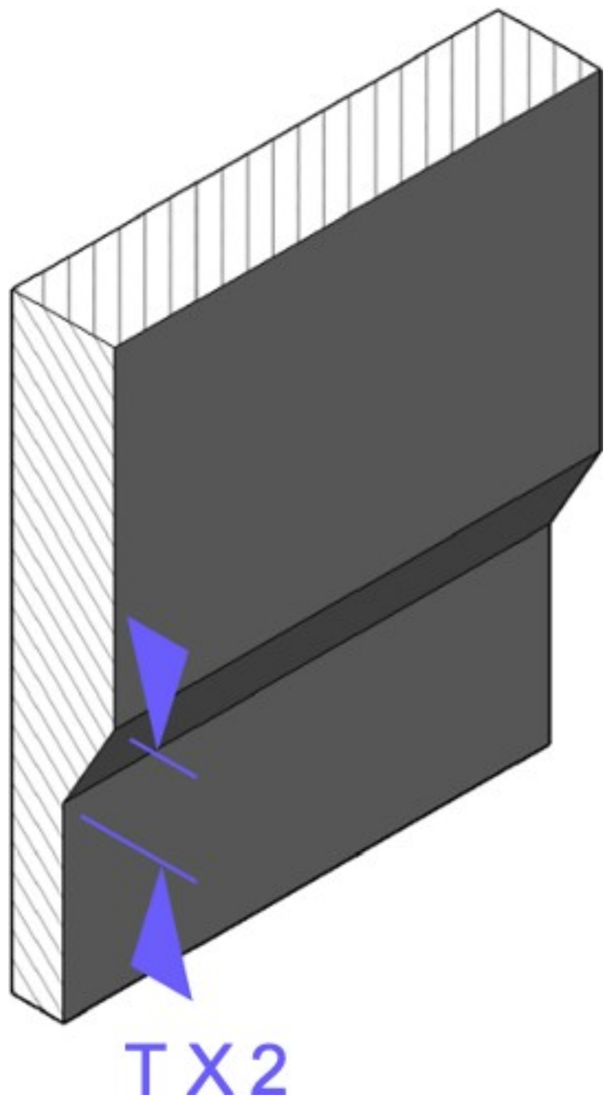
5.3 Gradual Transitions

Minimizing the rate of change from slice to slice dramatically reduces stress on parts during printing and minimizes the chances of warping, witness lines, and other defects. Rather than steps, use ramps, curves, fillets, or chamfers to create changes in part geometry.

Recommendation:

For a given rise T, the transition should run at least T x 2. Keep the maximum overhang angle of 40 degrees in mind because it may come into play depending on part orientation.





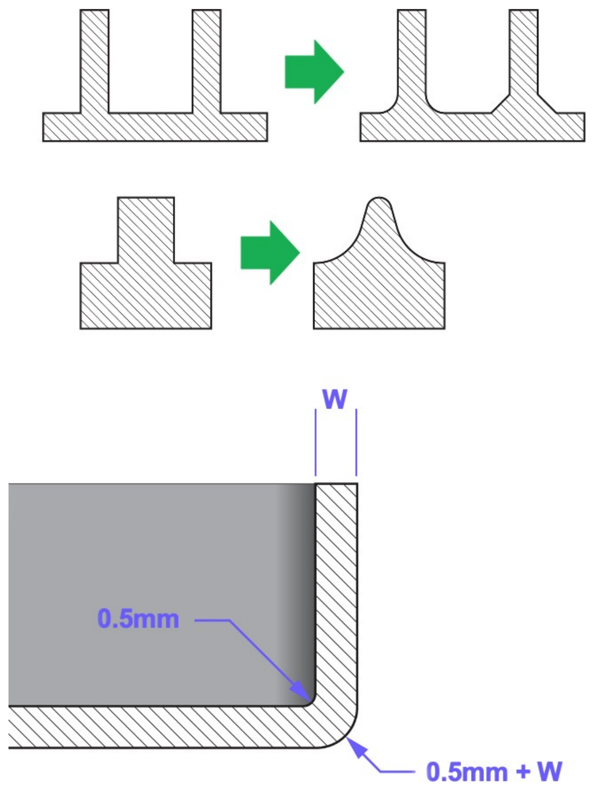
5.3.A Fillets and Chamfers

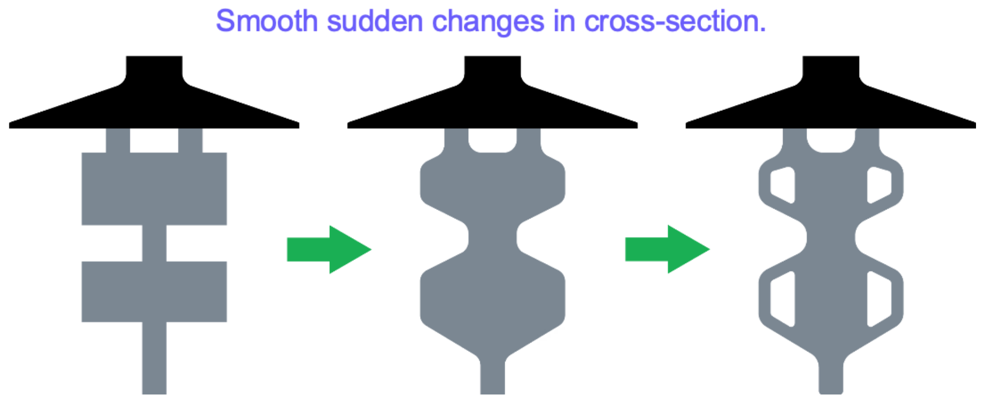
Sharp internal corners and sudden changes in a cross-sectional area from slice to slice create areas of high internal stress that are the most likely locations of print failure. While sharp corners tend to concentrate stresses, fillets (the rounding of an interior or exterior corner) distribute stresses over larger areas, leading to more durable parts. To lessen internal stresses, use fillets or chamfers instead of sharp corners by creating gradual transitions instead of steps.

Recommendation:

Interior corners ~ 0.05 mm minimum

Exterior corners ~ 0.5 mm + wall thickness





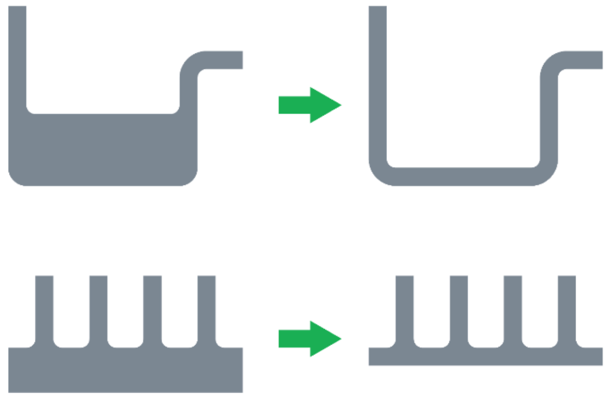
5.4 Wall Thickness

To keep material costs down for your part and to accelerate print speeds, walls should be as thin as possible while keeping in mind the minimum sizes for specific types of walls, which are outlined as follows.

5.4.A Uniform Wall Thickness

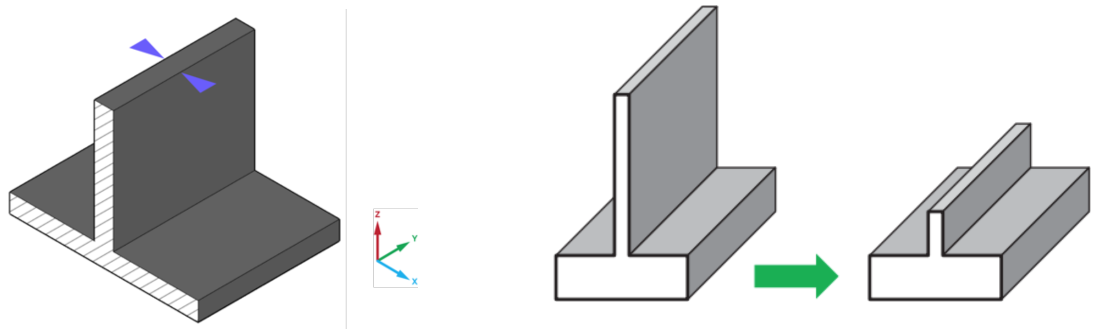
Part designs that maintain uniform wall thickness will minimize the effects of shrinkage and increase print success by allowing for equal shrinkage throughout the part. Keeping walls a consistent thickness can minimize warpage and will help avoid sudden changes in the cross section during printing.

Maintain consistent wall thickness.



5.4.B Unsupported Wall Thickness

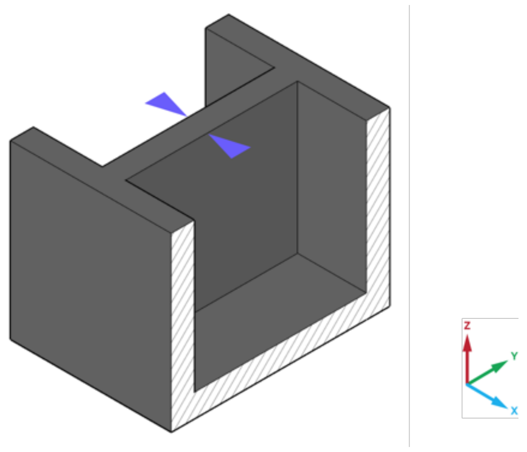
An unsupported wall is when the wall is only supported along one edge. These walls can be printed thinner than our listed minimums if you are prepared to iterate. When attempting unsupported walls at or below the minimum thickness, keep the walls as short as possible.



Unsupported Wall Thickness	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Recommended Minimum (mm)	2.5	2.5	2.5	2.5	2.5	2.5	2.5	2.5	2.5	2.5

5.4.C Supported Wall Thickness

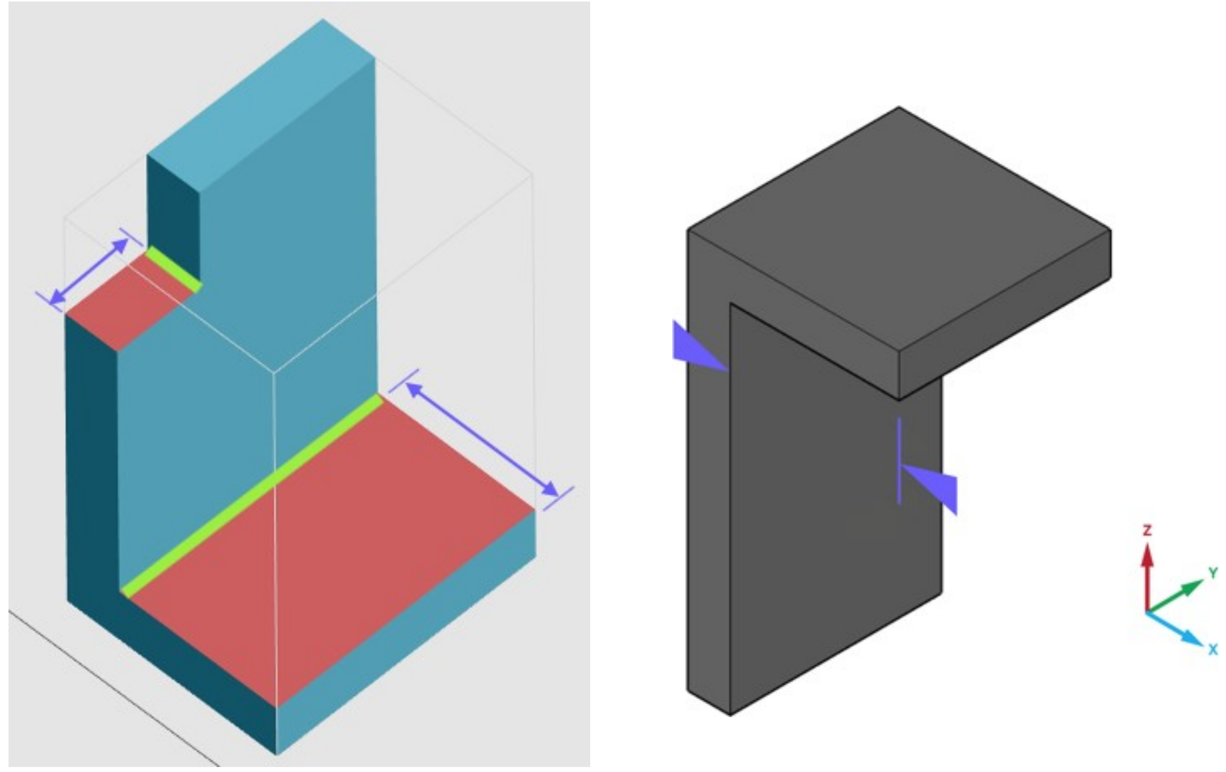
A supported wall is a wall connected to other walls along two or more edges. Supported walls can be made thinner because they are more stable during printing and less likely to warp during the washing process. As with unsupported walls, when attempting to design walls at or below the minimum thickness, keep the walls as short as possible.



Supported Wall Thickness	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Recommended Minimum (mm)	1.0	1.5	1.0	1.0	1.0	1.0	1.5	1.0	1.0	1.5

5.5 Overhangs

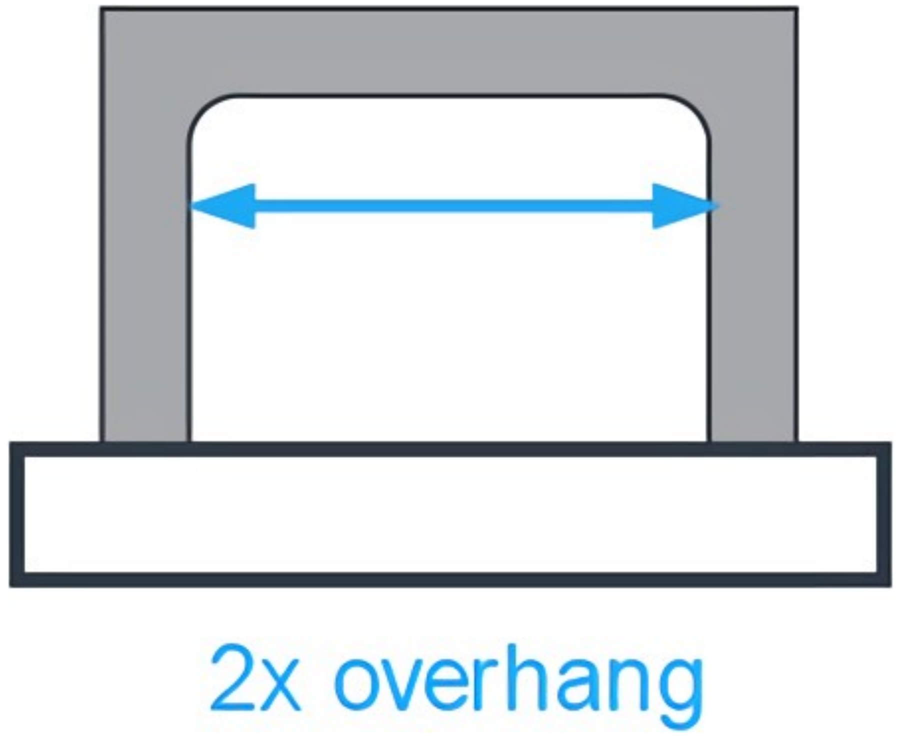
An overhang is an unsupported feature that projects from the model during printing. When compared to competing additive technologies, the Carbon DLS process allows much larger overhangs for your part. To ensure your overhangs will print successfully, measure the overhang from the edge where it meets the rest of the model and compare the value to the following table. Overhangs that approach maximum values may experience deflection during printing and therefore require supports or a redesign.



Overhangs	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Recommended Maximum (mm)	2.0	2.0	2.0	1.0	2.0	3.0	2.0	3.0	3.0	1.0

5.5.A Bridges

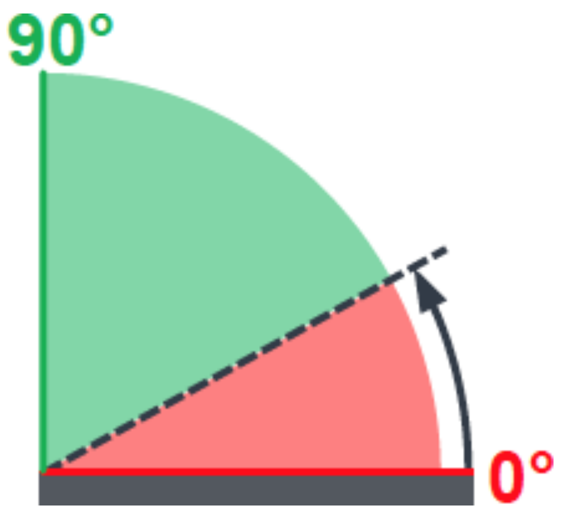
Bridges should span no more than twice the recommended overhang distance.



Bridges	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Recommended Maximum (mm)	4.0	4.0	6.0	2.0	4.0	6.0	4.0	6.0	6.0	2.0

5.6 Unsupported Angle

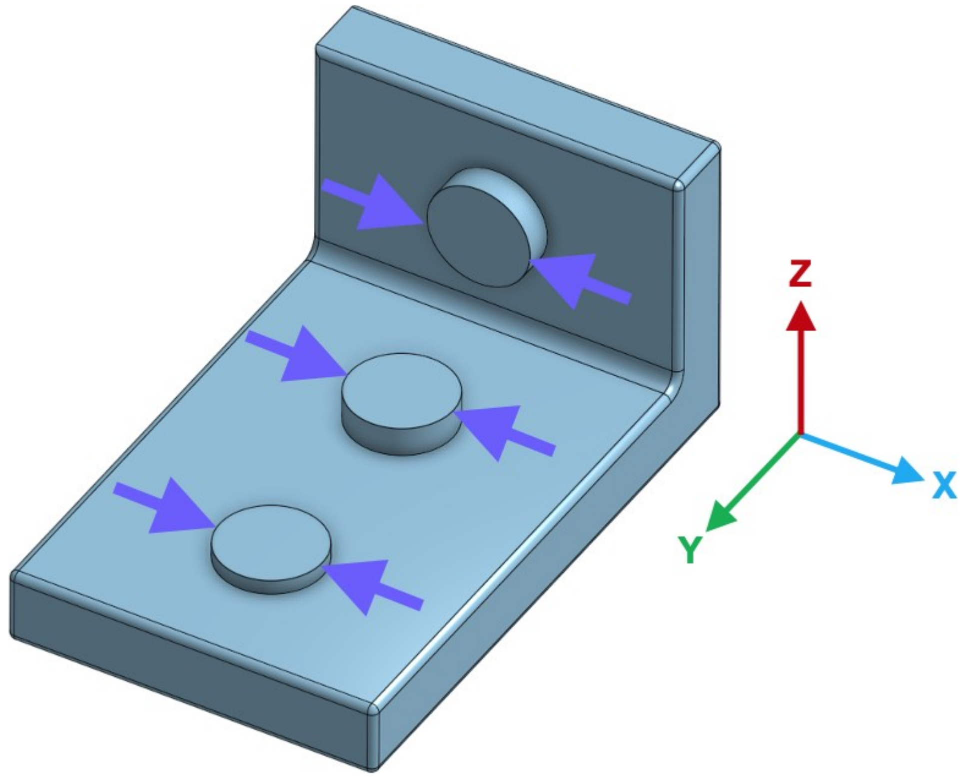
Unsupported angle refers to the angle that any unsupported feature is oriented relative to the platform (XY). The closer a feature is to being parallel with the platform, the greater the likelihood of failure. Conversely, the closer a feature is to vertical, the greater the likelihood of success. Thinner structures and more flexible materials may require more support or features oriented closer to vertical. In our print software, you can use the Overhang Detection tool to find potential trouble spots, but in general, overhangs less than 30 degrees should be avoided.



Unsupported Angles	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Horizontal (degree)	30	40	40	40	35	40	40	30	30	40

5.7 Positive Features

Positive features protrude from the surface of a part. We provide maximum size recommendations for positive features oriented horizontally in the XY print plane and vertically in the Z axis due to the different factors at play in each orientation. For feature orientations that differ significantly from either XY horizontal or Z vertical, use the XY values. Features smaller than our listed minimums can be attempted if you are prepared to iterate.

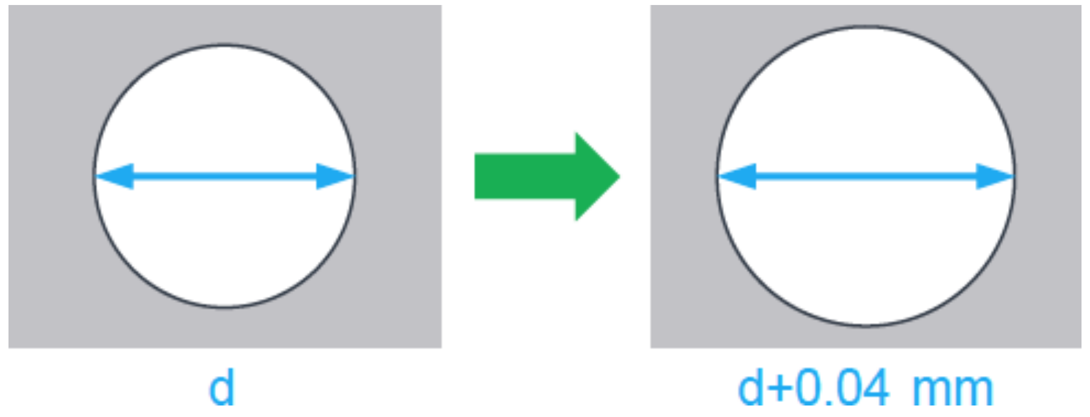


Positive Features	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Horizontal XY Recommended Maximum (mm)	0.4	0.3	0.4	0.5	0.5	0.4	0.3	0.6	0.4	1.0
Vertical Z Recommended Maximum (mm)	0.2	0.3	0.2	0.2	0.2	0.2	0.2	0.2	0.2	1.0

5.8 Holes

Similar to having to think about a mold's weld lines, undercuts, etc. as you design a part for injection molding, it's important to think about your part's orientation as you design holes (openings or features that pass completely through a part) that will be printed with the Carbon DLS process. For more on print orientation, go to section 6.2.

We provide minimum size recommendations for holes based on the part's orientation on the build platform: horizontally in the XY print plane or vertically in the Z axis. For hole orientations that differ significantly from either XY horizontal or Z vertical, use the XY values. To compensate for overcure, horizontal hole diameters should be oversized by ~0.04 mm. Holes smaller than our recommended minimums can be attempted if you are prepared to iterate.

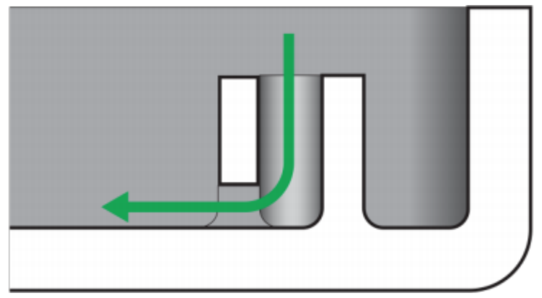


Oversize horizontal holes by ~ 0.04 mm to compensate for overcure

Holes	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Horizontal XY Recommended Minimum (mm)	0.5	0.5	0.9	0.5	0.5	1.0	0.6	0.9	0.9	2.0
Vertical Z Recommended Minimum (mm)	0.6	0.5	0.8	0.5	0.5	0.7	0.9	0.6	0.8	2.0

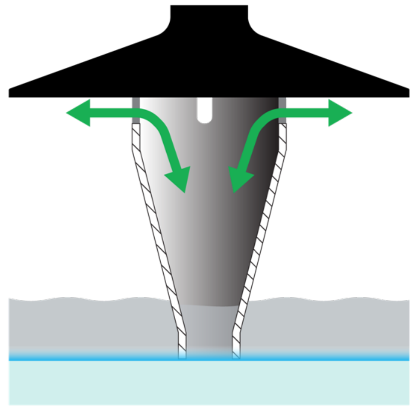
5.9 Blind Holes

Blind holes (e.g., bosses) only go to a specific depth and do not pass through a part. We recommend that you avoid blind holes if possible because they trap and hold resin during post-processing, requiring dedicated cleaning with swabs or other tools to remove it. In most cases, it is not possible to remove all of the uncured resin from the blind hole. If a hole with a set depth is necessary, add an opening to properly vent the part during printing.



5.10 Venting

Venting is essential for resin flow during printing; it also provides a clear path for liquid resin, solvent, and air to flow through during cleaning. Because insufficient venting is a common source of print failure, we recommend the addition of vent holes through redesign. Vent holes should be located at or near the build platform and should be a minimum of 2–3 mm wide.

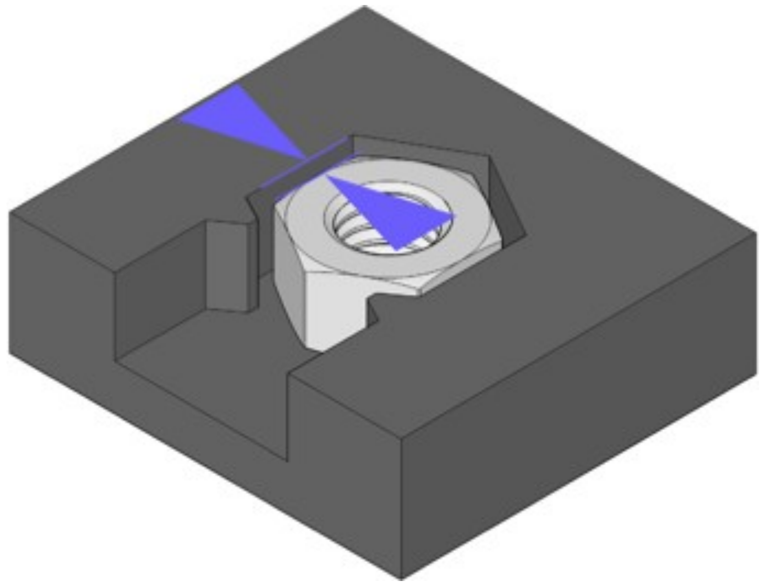


5.11 Metal Inserts

Following are a few ways to create metal inserts for parts printed with the Carbon DLS™ process. Note that while you are able to use thread-forming screws, they work better when printed from tough Carbon materials that can be successfully drilled into, like RPU or EPX. Conversely, stiff CE and elastic EPU would be poor material candidates for thread-forming screws.

5.11.A Nuts

The following figure shows a cavity designed to hold a hexagonal nut. The nut pocket resists the rotation of the inserted nut so that the screw easily threads into the nut, and should allow a small amount of play so that the screw can properly thread into the nut without stripping. Our recommended clearance for the nut is 0.25 mm.

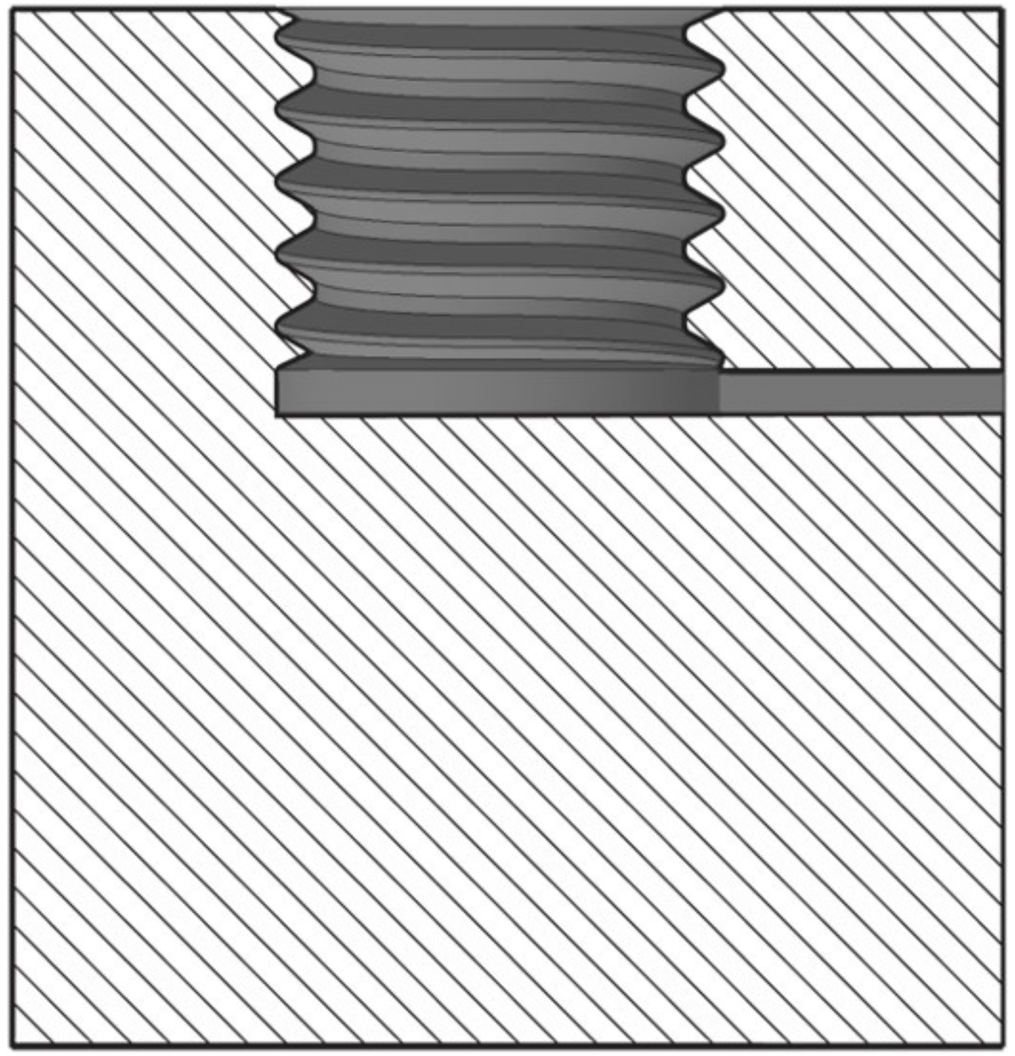


5.11.B Inserts

For coiled wire inserts that are screwed directly into the part, start by designing a hole 0.25 mm smaller than the manufacturer-recommended plastic hole diameter. After printing, ream out the hole with a drillbit of the same size as the manufacturer-recommended plastic hole diameter, carefully tap the hole, and insert the helicoil.

5.12 Threads

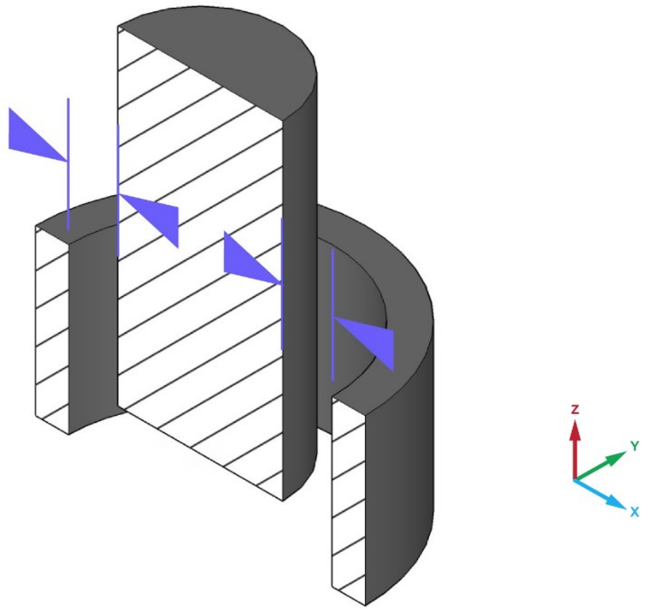
Our high-resolution process is capable of printing end-use threads. We recommend printing threads no smaller than M4 (0.7-mm pitch). Venting the hole is strongly encouraged to ensure the most accurate threads possible. Orient threads parallel to the platform for the best accuracy. After baking, chase the threads with an appropriately sized tap. Match the hole with a machine screw, not self-threading screws.

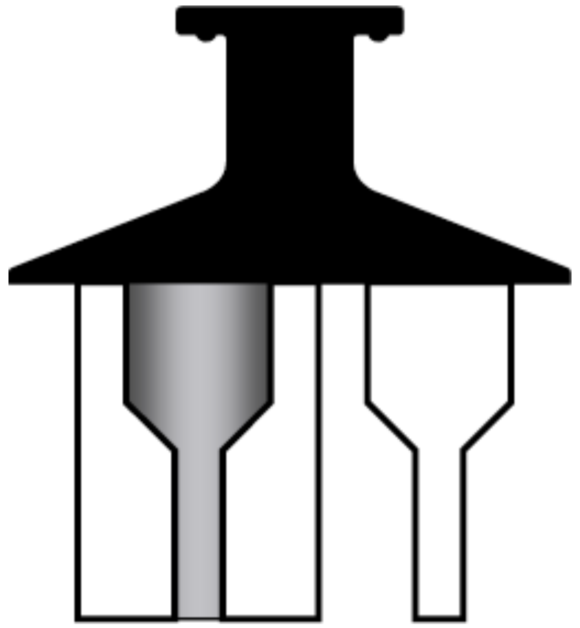


5.13 Clearance for Mating Parts

Mating parts are parts that fit together, such as a post that fits into a hole, a connector, an electrical housing, etc. When printing mating parts, follow the recommended clearance minimums between each part in the table below.

Because mating parts usually require the assembly of two or more components, it's important to account for variance throughout the production process, including printing and curing. The best way to reduce variance for mating parts and get the best fit is by printing them in the same orientation that they will be assembled.

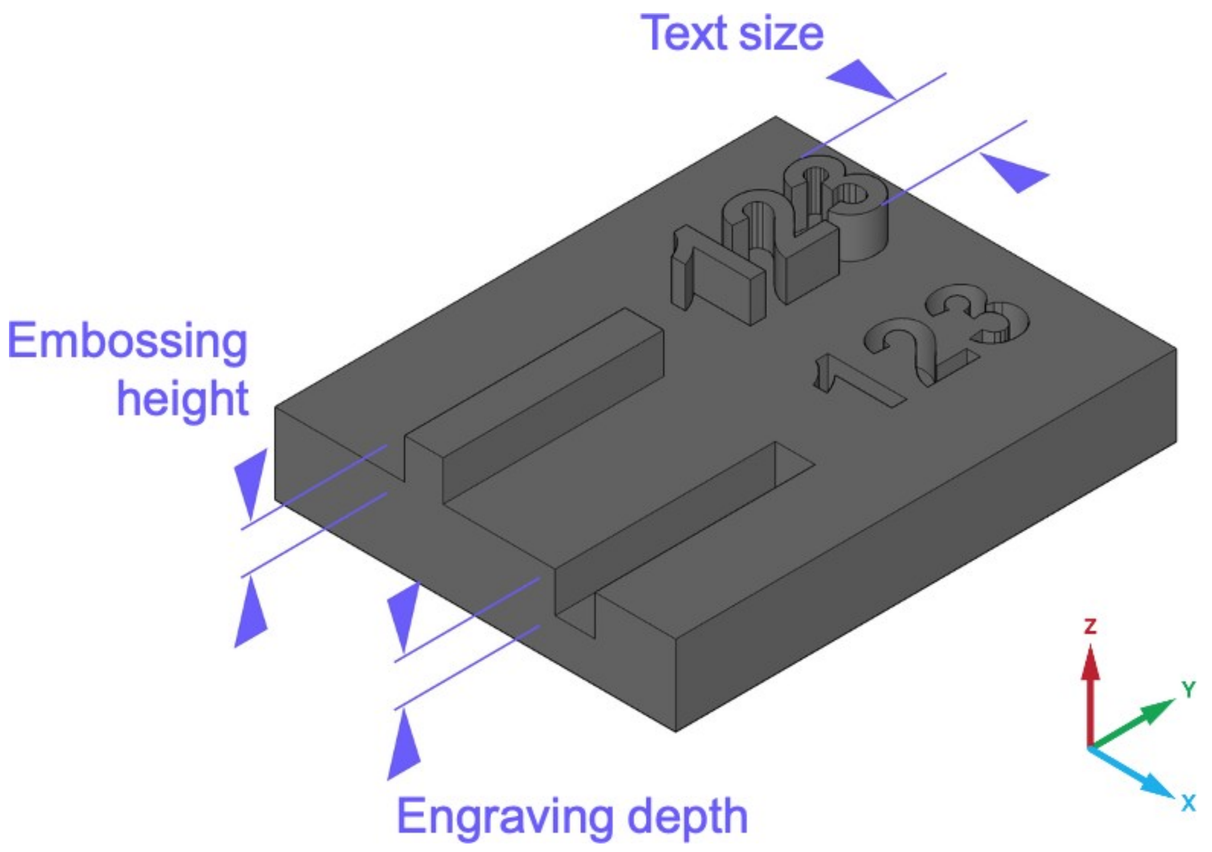




Clearance	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Recommended Minimum (mm)	0.4	0.5	0.5	0.5	0.5	0.8	0.4	0.5	0.5	0.5

5.14 Engraving, Embossing, and Text Sizes

Engraved text is etched into the part's surface, while embossed text protrudes from the surface of a part. Text size refers to the height of the letter and is the same for both engraved and embossed text. For maximum resolution quality, all text features should face the window (not the build platform) and follow the recommended minimums below.



Engraving and Embossing	RPU 70	RPU 130	MPU 100	EPU 40/41	FPU 50	CE 221	EPX 82	PR 25	UMA 90	SIL 30
Engraving Depth/Embossing Height Recommended Minimum (mm)	0.3	0.3	0.3	0.3	0.3	4.0	0.3	0.3	0.3	5.0
Text Size Recommended Minimum (mm)	3.0	3.0	3.0	3.0	3.0	3.0	3.0	3.0	3.0	3.5

6. Optimize for Printing

After utilizing the previous guidelines to design your part features for the Carbon DLS™ process, you can now focus on optimizing your part for printing, starting with support structures and build orientation. By strategically orienting your part and efficiently placing support structures, you can successfully optimize for printing with a specific goal in mind, whether that be reducing print time, cutting costs, accentuating aesthetics, or maintaining part functionality. In general, overhangs, unsupported angles, and the thickness of unsupported walls will inform the support strategy for your part. Note that there are limits to print optimizations due to external forces that affect Carbon DLS, which are explained further in the following sections.

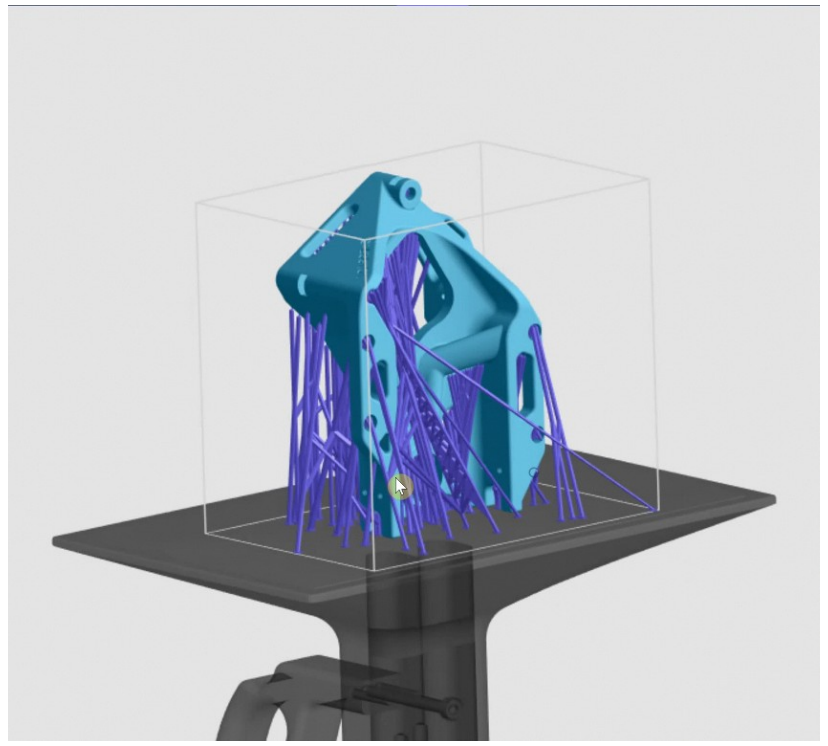
6.1 Support Structures

Supporting a part is critical to ensuring its mechanical integrity throughout the print. Support structures are made of the same material as the part and are generated to be easily removed by hand with minimal post-processing after the build is complete. Supports can be automatically or manually added through our print UI, or can be added through external design software. Support structures can be further optimized to reduce resin usage.

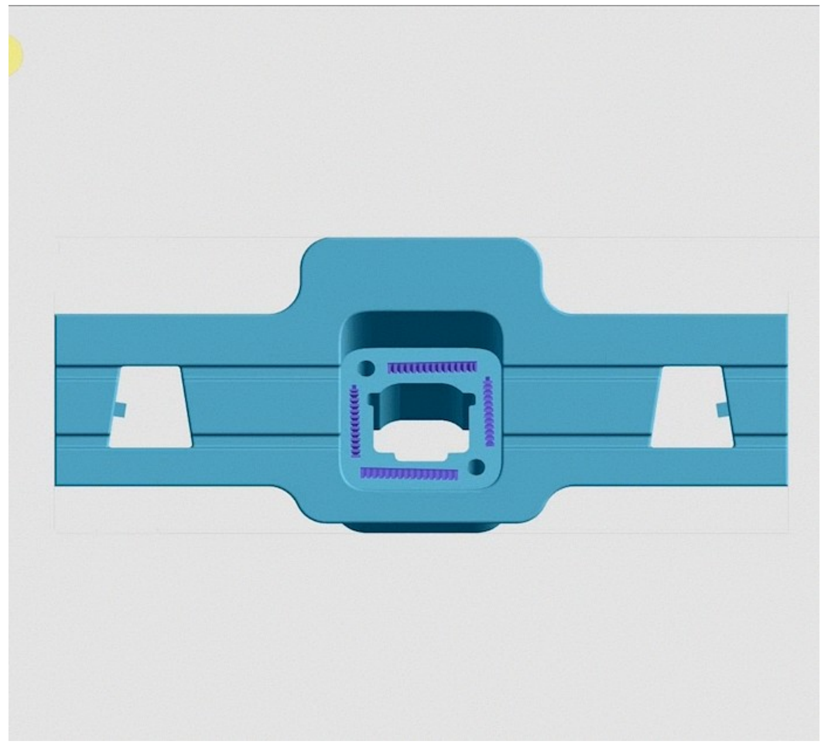
When adding automated supports to your part, you can choose between basic auto supports or advanced auto supports. If you are iterating the model/print design, Carbon recommends using basic auto supports because they are optimized to generate quickly (~5 minutes or less) while delivering a part that will print correctly in the majority of cases. These supports are calculated on the printer and take into account the material/resin type and the surfaces' orientations. While advanced auto supports take longer to generate (~90 minutes), they are more complete and appropriate for the specific material in use, its properties during the printing process, and the geometry of the part. These supports are calculated in the Carbon Cloud and offer better optimized placement based on the simulated forces on the part at every slice of the printing process.

6.1.A Support Artifacts

Areas with support structures may show raised bumps where the structure was removed. In certain materials, like EPX, SIL, and EPU, supported areas may be more visible because post-processing options are limited. This may affect areas like O-ring grooves and internal features if support is required in those regions. If your part emphasizes surface finish and cannot have support artifacts, either utilize fence supports (better for aesthetics) instead of bar supports, orient the surfaces so that supports will not be required, or design that area of the part to be support-free.



Bar supports (default)

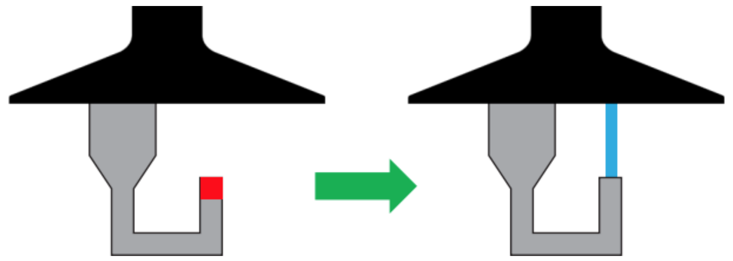


Fence supports

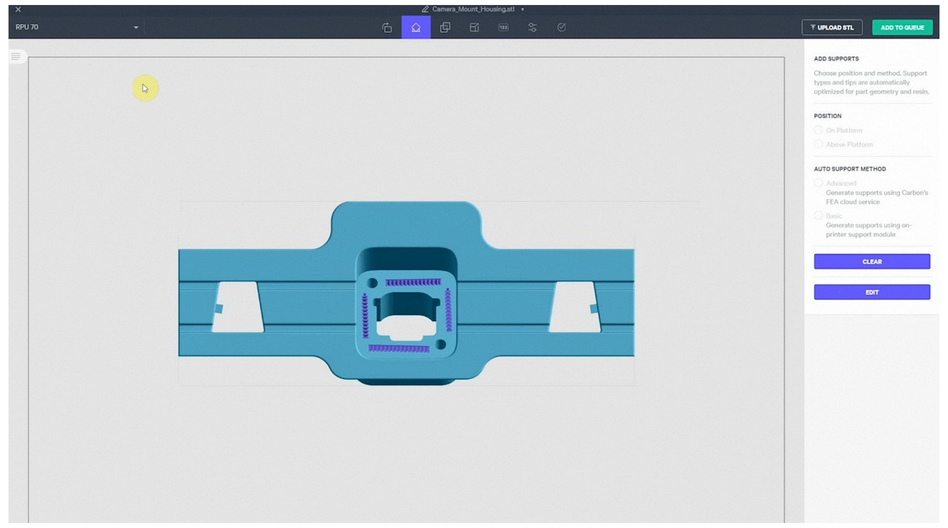
6.1.B Main Support Tips

- Adjust export settings to make a smooth model before you begin adding supports.

- Check overhangs and unsupported angles.
  - Use the Overhang Detection tool in the print software.
- Place supports no closer than the recommended overhang distances from part walls and other supports.
- Support any slice islands (unstable features that suddenly appear in the slice video) or redesign to attach to the part to prevent defects. Identify islands through the clipping plane software tool.



- Reinforce supports taller than 76 mm.
- Use advanced supports for a successful first print.



Choose position and method. Support types and tips are automatically optimized for part geometry and resin.

○ On Platform

○ Above Platform

○ Advanced
Generate supports using Carbon's FEA cloud service

○ Basic
Generate supports using on-printer support module

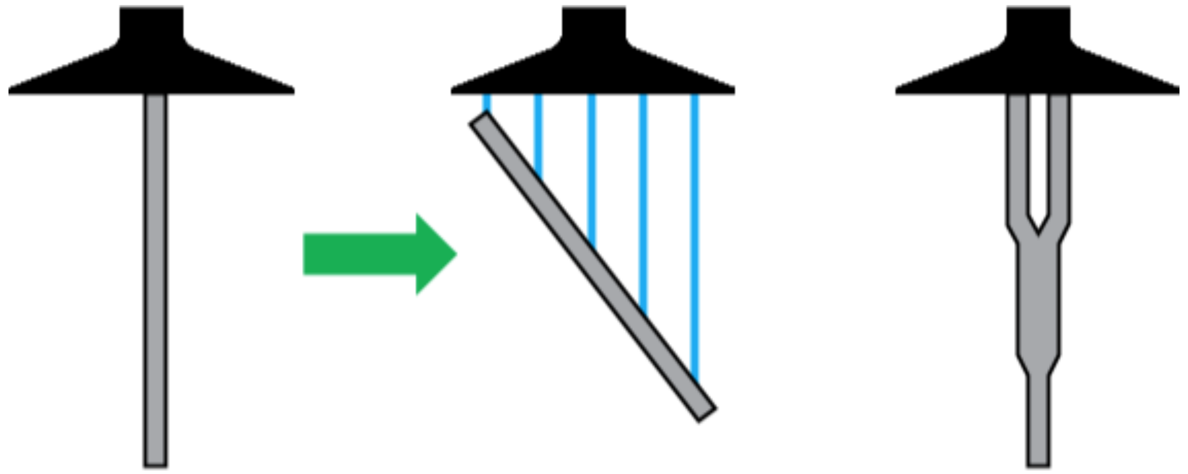
The supported part that is ready to print.

The supported part printing.

The printed and cleaned part.

6.2 Build Orientation and Print Layout

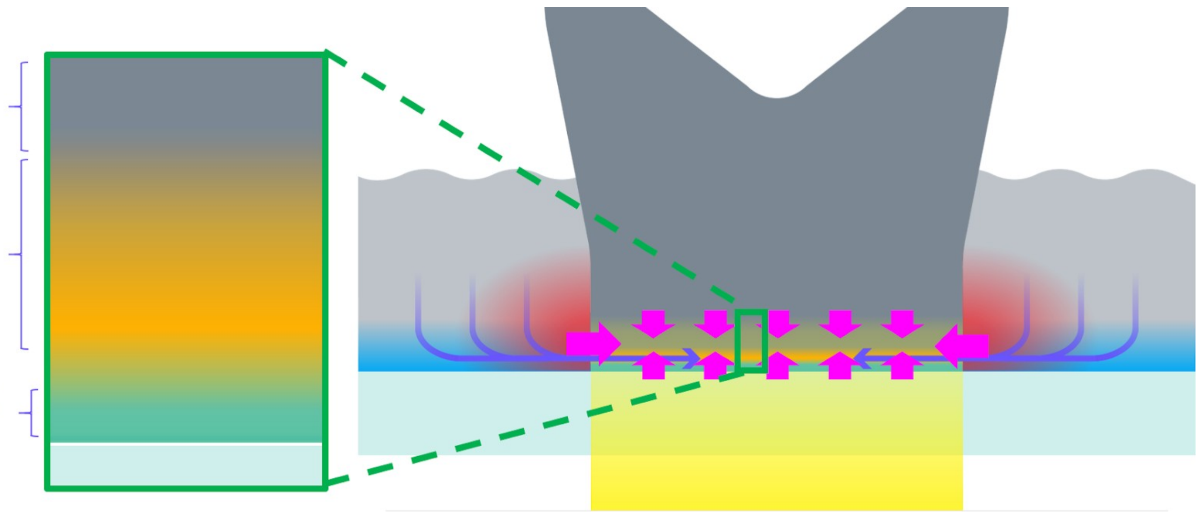
Build orientation will not affect the mechanical properties of the part but will dictate how and where support structures are used. Orient your build to optimize prominent features or details while simultaneously minimizing build height (Z axis) and support usage. Adding additional supports can be justified if it helps effectively reduce the chance of failures during the build. For example, to avoid warpage with tall, thin parts, either change the orientation and add supports, or redesign the part to reduce height and create stability. Additionally, parts with large or broad flat sections should be oriented at a minimum of 15–20 degrees from the platform to minimize the cross section per any given layer position.



To maximize the number of parts per build, use the minimize footprint automated software tool to orient the part to occupy the least amount of surface area on the build platform. Then use the auto layout and automatic padding features to arrange the parts evenly across the platform, saving you significant time as you set up your print. This combination of automated software tools can be leveraged for printing duplicate parts or multiples of different parts, which is especially important when producing sets of parts. Because of the thermal limitations (see section 6.3.A) of the Carbon DLS process, it's important not to overpack your build when optimizing your print layout.

6.3 Factors That Affect Your Print

The chemical reactions that solidify the resin generate heat, and the amount of heat generated is based on the amount of material being cured. Our printer software automatically takes this into account and optimizes printer speed to prevent overheating, keeping operating temperatures within acceptable limits. It is important to note that even after a part is fully optimized, heat will persist as the main limiting factor for how much you can reduce your print time without risking part and equipment damage. To reduce heat generation and maintain optimal print speeds, we recommend that part designs contain minimal cross sections. Refer to the preceding design guidelines for cross-section sizing specifics.



This gradient ensures isotropic properties by enabling robust entanglement between slices. **Each new slice bonds to its predecessors to form a uniform whole.**

The dead zone also creates a gradient of polymerization from less cure close to the window to more cure further from the window.

The oxygen-rich dead zone prevents curing just above the window. This keeps the part from sticking to the window and allows liquid resin to flow freely into the build area.

6.3.A Heat Generation – Thermal Limit

The chemical reactions that solidify the resin generate heat, and the amount of heat generated is based on the amount of material being cured. Our printer software automatically takes this into account and optimizes printer speed to prevent overheating, keeping operating temperatures within acceptable limits. It is important to note that even after a part is fully optimized, heat will persist as the main limiting factor for how much you can reduce your print time without risking part and equipment damage. To reduce heat generation and maintain optimal print speeds, we recommend that part designs contain minimal cross sections. Refer to the preceding design guidelines for cross-section sizing specifics.

6.3.B Suction – Resin Flow

During the printing process, the build platform is gradually pulled up to allow space for the part to build. This upward motion of the platform and part creates a suction force, which benefits the printing process by pulling fresh resin into the build area.

However, this suction force also pulls down on the part, potentially causing deflection, deformation, or detachment from the build platform. Ways to combat suction include:

- Avoiding designing parts with large cross sections, because larger cross sections can generate greater suction forces.
- Securing overhangs and stabilizing asymmetrical parts with supports.
- Redesigning a part to include a gusset-like feature that resists deflections and gradually increases the cross section of the part.
- Maintaining a relatively uniform cross section over the length of a part so the suction force cannot overcome the adhesion force and detach your part from the build platform.
- Orient the part's center of gravity as close to the platform as possible.

6.4 Specific Optimizations

I want to...

6.4.A Reduce Print Time

How?

- **Reduce Z height length.** The longer the Z height, the longer the overall print time (with EPU as an exception).
- **Minimize cross sections and wall thickness.** Large cross sections and thick walls utilize more resin and therefore impair resin flow, which delays the print so resin can flow across the window and underneath the build.
- **Increase spacing between parts.** This slows heat generation and reduces build delays caused by thermal limits.
- **Add ample vent holes.** This will reduce trapped volumes and therefore reduce build delays driven by allowing internal pressure and excess resin to settle.
- **Improve the support strategy.** Have you over supported your part? Supports take time to build just as your part does, meaning the more supports, the longer the print time. More supports also make a part harder to clean, which lengthens your post-processing.

6.4.B Minimize Cost

How?

- **Lightweight your part.** This will allow you to use as little material as possible.
- Improve the manufacturing efficiency of the build:
    - **Maximize throughput by packing the platform.** Produce more parts per build in nearly the same amount of time by using the software auto-layout and automatic padding features.
    - **Improve the support strategy.** Over-supported parts cost more (more material used), are harder to clean, and take longer to print. Can the support strategy be changed to improve any of these factors?
    - **Evaluate the cleanability of the build.** Will this build clean well in the part washer? Will each part need to be cleaned by hand? Reducing post-processing to a single process saves you both time and money.

6.4.C Accentuate Aesthetics

6.4.C.A Surface Finish

Because stepping is not present, parts will typically have a matte to semi-gloss surface finish depending on feature direction. What may look to be stepping is actually optical scattering known as the Moire pattern (see section 4.5.A).

How?

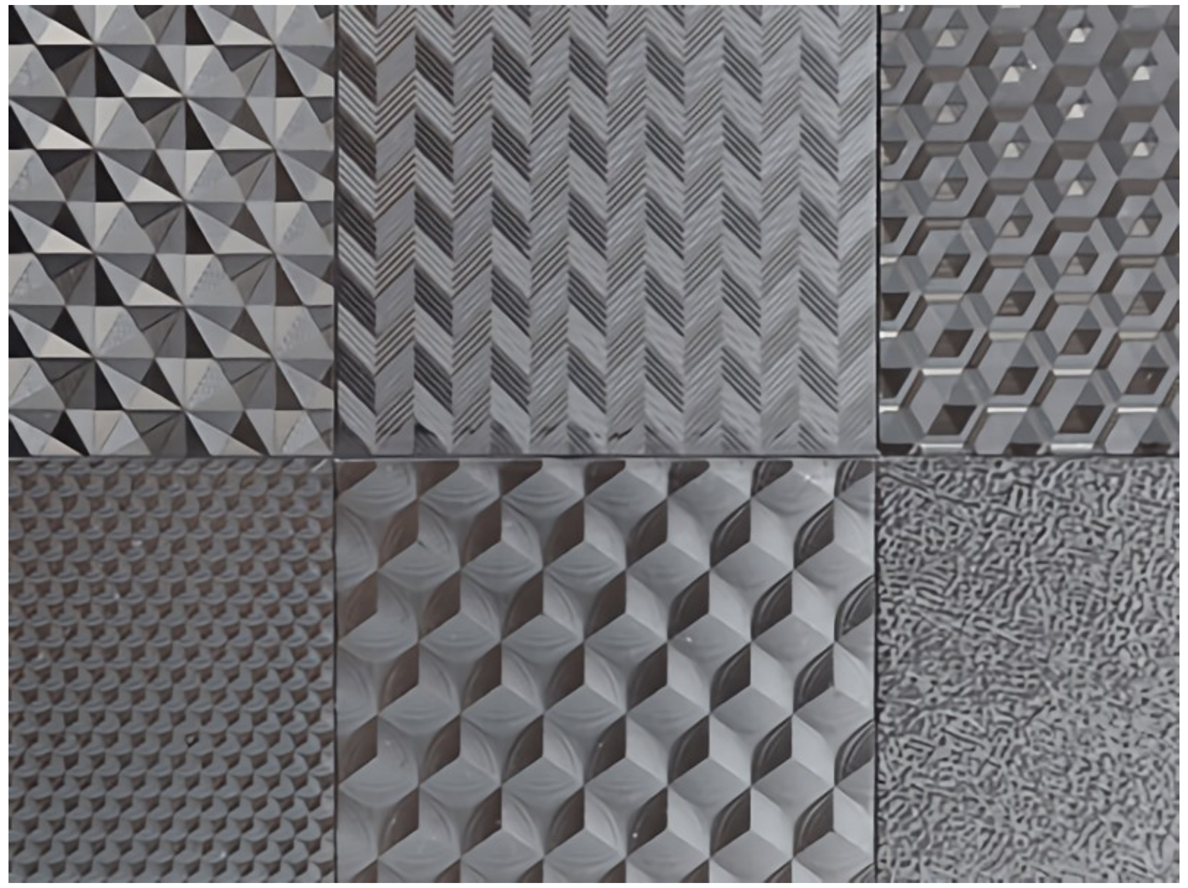
- Avoid using supports if possible.
- Print the consumer-facing side directly against the platform.

6.4.C.B Textures

Textures can be applied in CAD using various design software. Because of the continuous printing process, matte and geometric textures can be aesthetically pleasing and help mitigate cosmetic variations on the surface.

How?

- Do not apply the texture to every surface–the face against the platform should not have the texture.
- Reduce slice thickness from standard resolution (100 µm) to fine (25 µm). This will, however, increase print time.



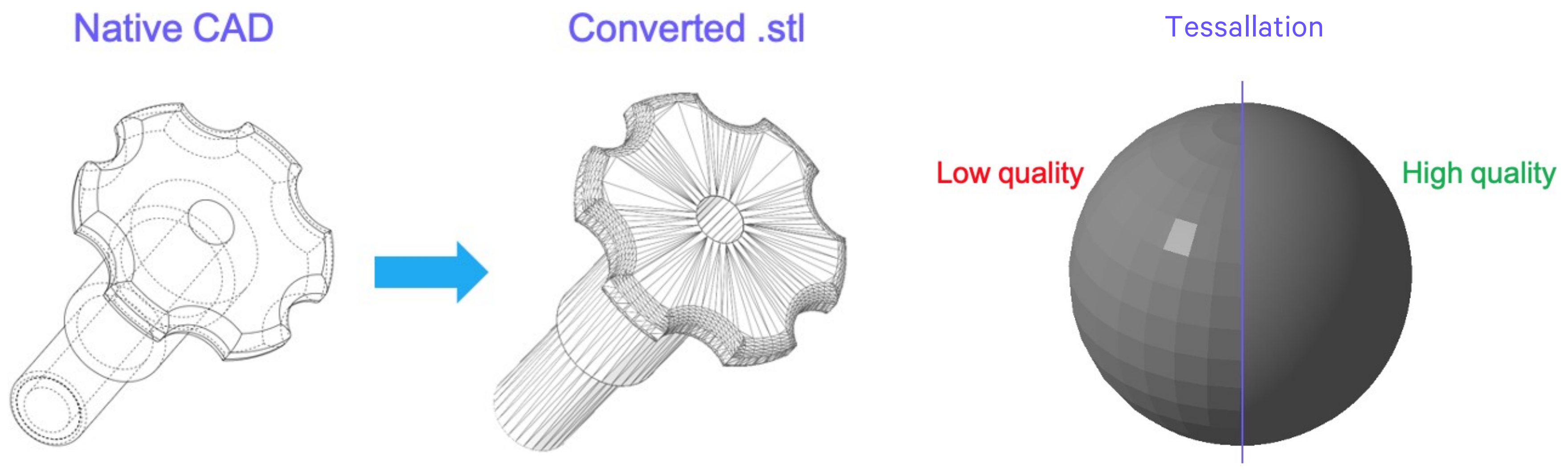
Variety of surface finishes available with the Carbon DLS™ process.

7. Preparing to Print

7.1 Model Tessellation

Before printing, digital models need to be exported from the CAD software where they were created to the .stl file format. The conversion process tessellates the model, converting it to an explicitly defined mesh of triangles usable by our printers. Proper tessellation will create a high quality model with smooth surfaces and well articulated features without faceting.

High quality digital models are necessary to maximize part quality. The Carbon DLS™ process is especially sensitive to the smoothness of digital models. Faceting that is not visible in other additive processes can be seen on parts printed with the Carbon DLS process.



7.2 Exporting CAD Models

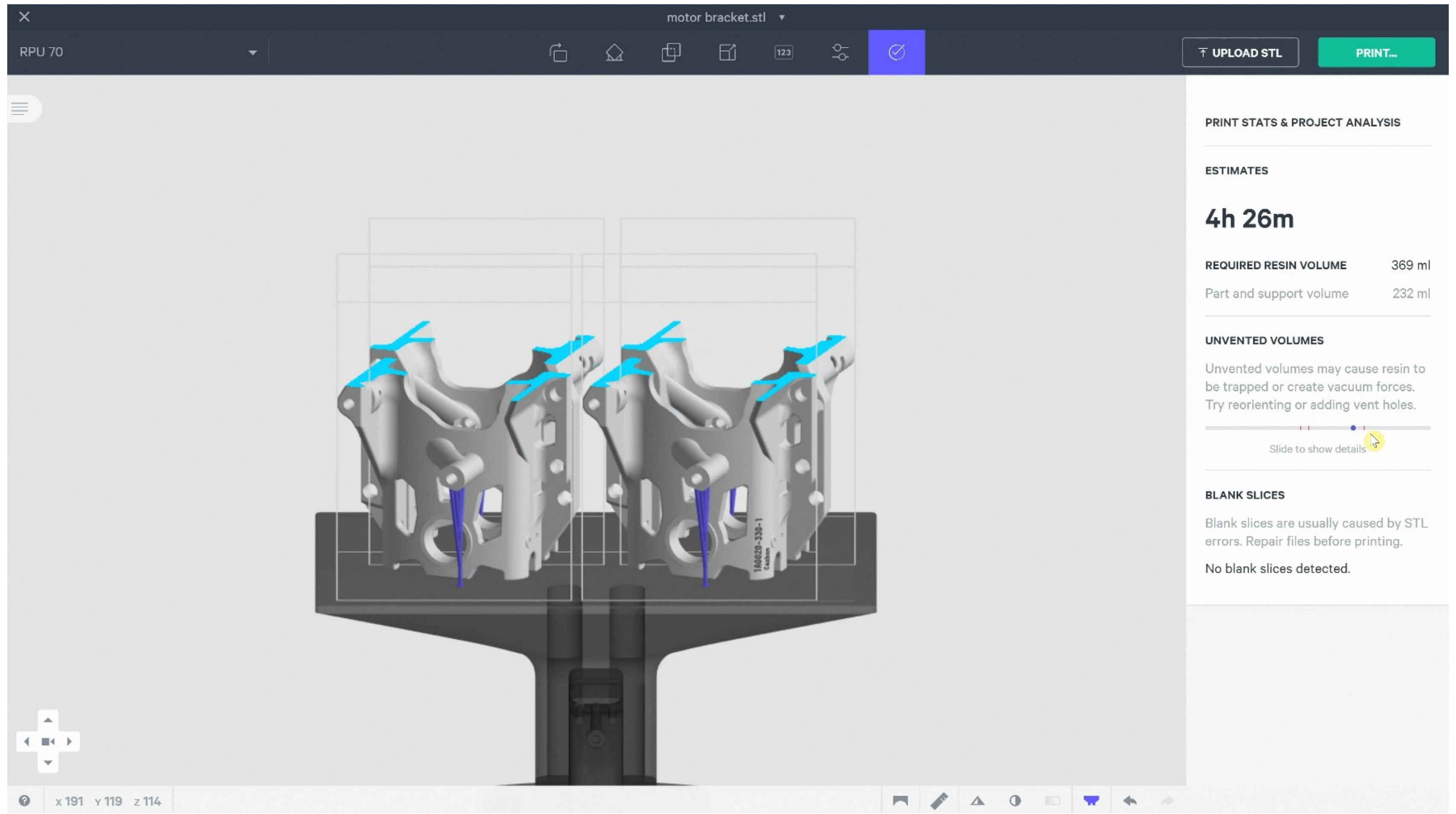
Most CAD programs offer options to optimize the tessellation of parts before exporting. These options are typically surface deviation and angle. Smaller values that match the resolution of our printers create smoother geometry. You do not need to go lower than the following values.

Recommended tessellation values:

- Surface deviation = 0.01 mm
- Angle = 1.0 deg

7.3 Final Print Analysis

After you finish supporting your part and are about to print, our software will analyze your project to provide you with important print statistics including print time, required resin volume, and part and support volume. This analysis also detects unsupported areas and unvented volumes, and issues prompts to fix those potential problem areas ahead of time so you can print successfully. While the print analysis is helpful, do not rely on it to fully troubleshoot your part—it's still crucial to implement the preceding design and print optimization guidelines before getting to this step.



PRINT STATS & PROJECT ANALYSIS

ESTIMATES

4h 26m

REQUIRED RESIN VOLUME 369 ml

Part and support volume 232 ml

UNVENTED VOLUMES

Unvented volumes may cause resin to be trapped or create vacuum forces. Try reorienting or adding vent holes.

Slide to show details

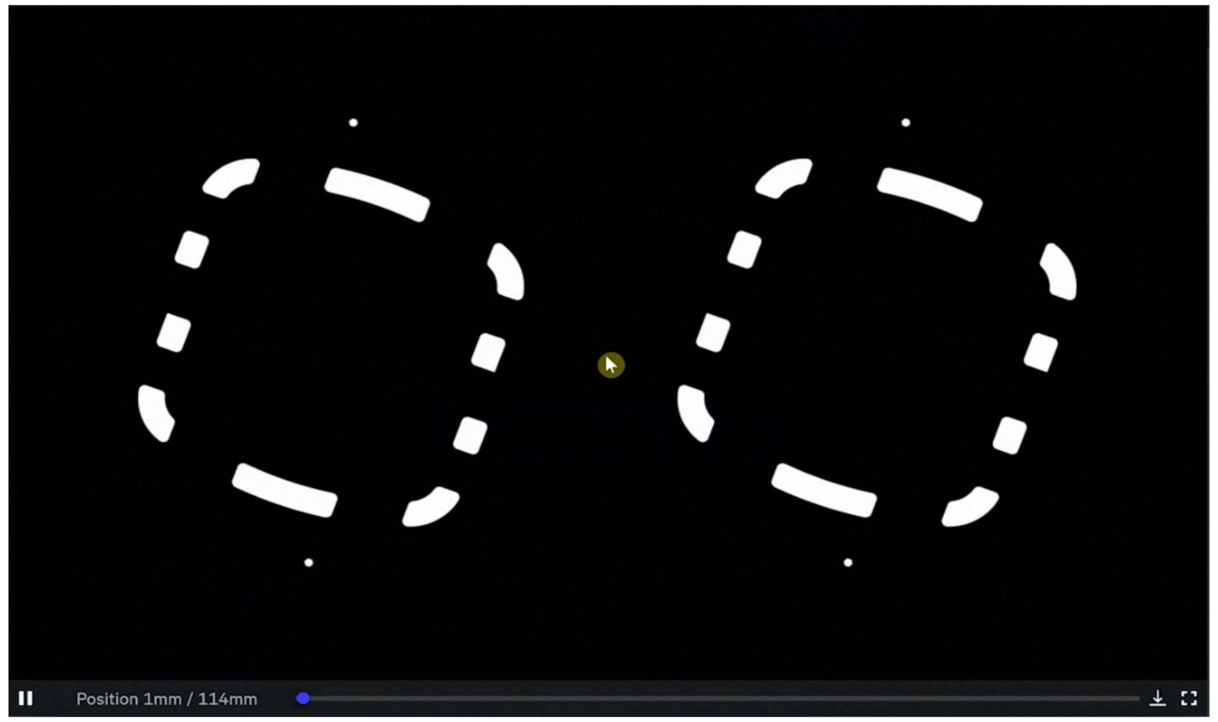
BLANK SLICES

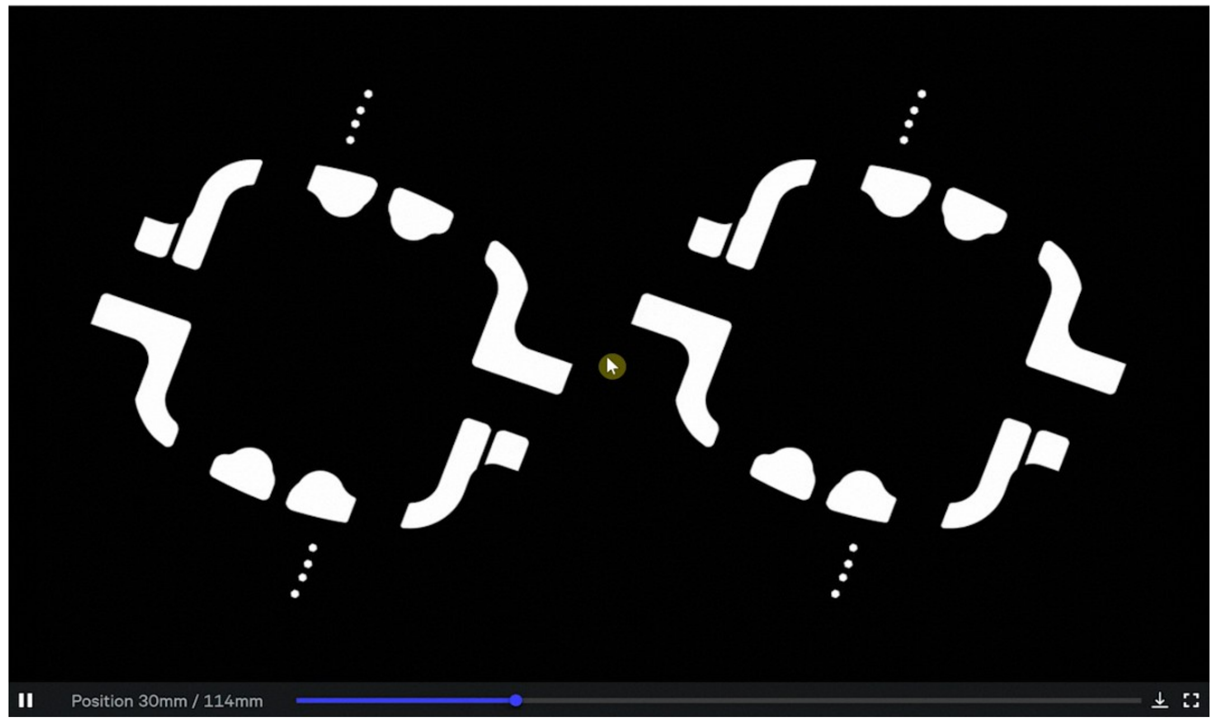
Blank slices are usually caused by STL errors. Repair files before printing.

No blank slices detected.

7.4 Slice Movie

Last, the software examines the slices that comprise the geometry of your part via a slice movie. The white areas show where UV light will be projected in each slice–this is where liquid resin will cure and your part will solidify. When watching this slice movie, be on the lookout for empty slices (entirely black frames)–this means nothing will print during that particular slice, which will likely cause your print to fail.





You are now ready to print!

8. Developing Standards and Specifications

8.1 Earlier Process Monitoring and Part Testing

While there can be sources of variation (machine-to-machine, printer-to-printer, batch-to-batch, etc.) in your print, a major advantage of the Carbon DLS™ process is the ability to conduct extensive process variation testing early on, because prototyping and production occur on the same platform. You can begin part testing much earlier in the product development cycle, which is not something that can typically be done using traditional manufacturing methods without incurring significant additional costs. The ability to test multiple, functional variations allows you to analyze your performance spread, letting you understand where the variation is, optimize quickly through rapid design iterations, and ultimately speed up your time-to-market.

We offer many opportunities to incorporate modern digital process-improvement techniques by providing network-connected monitoring at the part level, the machine level, and the fleet level. These features become particularly important if you're considering customization.

8.2 Writing Specifications for Production Parts

Because additive manufacturing in general is still a relatively new process for production parts, many companies and industries have not yet written standard requirements for 3D printed parts.

Before you start writing specs for your part, ask yourself: is this process predictable and is its performance repeatable? If the performance spread meets your part's needs, you can begin writing specifications for the process to establish a PPAP (production part approval process).

With a dialed-in process that demonstrates reliable and consistent production of parts at spec and within tolerances, you have the opportunity to continue perfecting your part through an optimization approval process and to seamlessly scale production.

9. Internal Adoption of Carbon DLS™

The adoption of any additive manufacturing technology is a journey for not just industries at large but also for individual teams. Here are some tips on how to accelerate the adoption of the Carbon DLS process in your company and team:

1. Unlearn traditional design limitations and lean into the design freedom of the Carbon DLS process.

- Share successfully printed parts and circulate this guide and the Carbon DLS process design guidelines as resources and reminders.
- Host design and/or application discovery competitions to experience new design freedom and encourage out-of-the-box design ingenuity among teammates.
- Engage outside experts for help getting started. Many contract manufacturers like those in the Carbon Production Network have extensive experience identifying applications, developing designs for 3D printing, and running serial production at small and large scales.

2. Understand the pain points that Carbon DLS can help overcome.

- Share application discovery resources among teammates so they can start to think more critically about what types of applications can best leverage the benefits of the Carbon DLS process.

3. Clarify and align around what the Carbon DLS process can and cannot do for your team.

- The Carbon DLS process cannot do everything. Be honest ahead of time and help your colleagues understand both what it can and can't do.
- The Carbon DLS process doesn't have to require radical reinvention; in many of its most successful applications, it works alongside conventional methods. Whether you have an in-house Carbon printer or work through a contract manufacturer, position the Carbon DLS process as a complement to traditional manufacturing technologies, showcasing applications like jigs, fixtures, and bridge tooling.
- Discuss the realistic current potential and the future potential of Carbon DLS for your team and how you can incorporate it into your business strategy.

Conclusion

We hope this guide gives you a deeper understanding of the Carbon DLS™ process and has inspired you to lean into the design freedom and economic advantages of 3D printing. Whether your goal is one part or one million custom pieces, you are now equipped with the knowledge, tools, and best practices to design better parts for your product and achieve the best results with the Carbon DLS process.

Request Sample Parts

Interested in experiencing parts printed with the Carbon DLS™ process? Visit our "Get Parts" page to request sample parts.

Our free engineering kit includes an elastomeric lattice puck made of EPU 41 and three tensile bars made of our versatile rigid polyurethane materials, RPU 130 and RPU 70, and our epoxy-based material EPX 82.

GET THE FREE ENGINEERING KIT

Get a Part Made

Do you have a part or prototype you would like printed with the Carbon DLS™ process? Tell us more about your part needs here and we will connect you with the right manufacturer from our global network of production partners.

GET A PART MADE

Carbon®

3D as It’s Meant to Be