3D Printing for Battlebots

Table of Contents

→ Additive Manufacturing Basics

Additive manufacturing (AM): the process of manufacturing a three-dimensional object from a CAD model or other digital model, layer by layer; commonly known as 3D printing.

Common Uses of Additive Manufacturing

- → **Rapid prototyping:** quick iteration makes it easy to test many design options and reach a better design faster
- → **Part consolidation:** parts can be combined without increasing manufacturing complexity; fewer parts means fewer potential failure points
- → **Optimization:** combining 3D printing with computational design (e.g., generative design or topology optimization) can make parts stronger and lighter
- → **Customization/personalization:** parts can be modified and manufactured for varied geometries, conditions and preferences

Additive Manufacturing Processes

According to ASTM F2792-12, there are seven 3D printing processes:

- Material extrusion
- Vat photopolymerization
- Powder bed fusion
- Binder jetting
- Material jetting
- Directed energy deposition
- Sheet lamination

The most common processes at the amateur/hobbyist level are material extrusion and vat photopolymerization. Both are available at makerspaces around campus. The other processes require more complicated equipment or more expensive materials or are mostly used for industry-specific applications. Powder bed fusion is the third most common process, and there are several machines around campus capable of printing in polymers and metals. However, there are very few that are available for general use. Material jetting is also available on campus, but it is expensive and has few Battlebots applications.

Material Extrusion

Material extrusion refers to technologies that involve pushing material through a heated nozzle to be deposited layer by layer. The most common technology, and the most popular 3D printing technology overall, is fused deposition modeling (FDM), also known as fused filament fabrication (FFF). In this case, the raw material is thermoplastic filament. The Prusa i3 MK3S+, Ultimaker S3, and Creality Ender 3 are all examples of FDM/FFF printers.

Advantages and Disadvantages of FDM/FFF

Advantages	Disadvantages
• Quick and cheap • Thermoplastics have a wide range of properties • Easy to use • Multi-material possibilities	• Produces noticeably anisotropic parts • Relatively low resolution and accuracy

Vat Photopolymerization

Vat photopolymerization, colloquially known as resin printing, uses a light source to harden photosensitive liquid resin layer by layer. The most common technology is stereolithography (SLA). In SLA, a laser moves across each layer column by column. Formlabs Form products like the Form 3 are examples of SLA printers. Another common technology is digital light processing (DLP), which uses a light source and tiny, rotating mirrors to cure each layer all at once; it is much faster but usually more expensive than SLA. Masked stereolithography (MSLA) is a cheaper alternative to DLP. Instead of tiny mirrors, the light source sits under an LCD screen that acts similarly to a flashlight shining through cutouts that change with each layer.

Advantages and Disadvantages of SLA

Advantages	**Disadvantages**
• High resolution allows for fine details • Relatively dimensionally accurate • Produces isotropic parts	• Resins are messy and often toxic • Usually requires multiple post-processing steps (alcohol wash, secondary UV cure, surface finishing)

→ Design for Additive Manufacturing

General DfAM Considerations

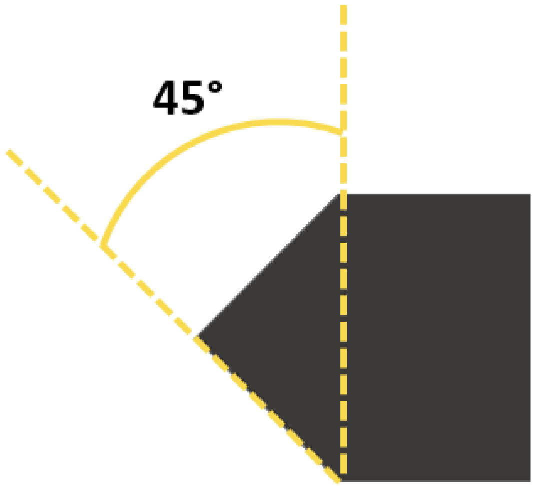
- → Manufacturing process characteristics, including support requirements, anisotropy, and resolution (see Slicing & Settings)
- → Material options, mechanical/thermal properties, and post-processing requirements (see Common Materials)
- → Application of AM - is it necessary/useful to 3D print this vs. using a conventional manufacturing technique based on time, cost, and part complexity?

Material Extrusion (FDM/FFF)

Process Considerations

Due to the nature of the layer-to-layer fusion, parts printed with FDM/FFF are anisotropic, meaning their mechanical properties differ depending on the orientation. In general, the material strength is higher than the strength of the layer adhesion. Parts are stronger in tension in the direction parallel to the layers but stronger in compression and shear in the direction perpendicular to the layers.

The melting and cooling process during printing results in parts that are usually larger than nominal dimensions with holes and other cut features that are smaller. The necessary tolerance varies enough by material and temperature that a small test print is always a safe option. A good starting point for tolerancing mating features is to increase (for cut features) or decrease (for extruded features) all dimensions by 1-3% in the printer's XY plane in conjunction with fit tolerances.



Overhanging features at angles more than 45° from vertical may require support, though some printers/materials can go up to 60°. Horizontal holes and channels smaller than ⅜" diameter usually do not need support material, but shrinkage of horizontal holes and internal cavities tends to be more significant in the direction perpendicular to the layers than in other directions.

Materials Considerations

Thermoplastic polymers are ductile and have good impact resistance. The rigidity and density of individual parts depend heavily on the infill and other print settings (see Slicing & Settings). One important consideration for Battlebots applications is the glass transition temperature (T_g), at which the material softens. If a FDM/FFF part is in contact with electronics, be mindful of overheating from extended use.

Vat Photopolymerization (SLA)

Process Considerations

From a design perspective, SLA has fewer process considerations than FDM/FFF, mostly because the parts are effectively isotropic, meaning their mechanical properties are independent of direction. One important consideration is the warping that comes from the curing process. Small cross-sections are more susceptible to warping so require more support to print correctly. Overhanging features at angles greater than about 45° from vertical need supports to print properly. SLA supports are tree-like and often leave bumps or small divots once removed, so if these surfaces need to be planar, post-processing like sanding or sand blasting is needed.

In general, SLA is capable of creating more complex geometry than FDM/FFF. Blocky parts or parts with large features are not usually a good application of the technology. If not designed carefully, however, intricate geometries will need more support and are more likely to fail.

Material Considerations

Acrylate resins like those most commonly used in SLA are thermoset polymers, meaning they experience brittle fracture, so once they yield, they're done. Even elastic materials like Formlabs Elastic or Flexible will fail before extensive plastic deformation. The nature of the base material and the printing process means that there is a wide range of mechanical, thermal, and electrical properties available. Different materials will have different support and post-processing requirements, so if interested in a particular resin, check that your part will print successfully before finalizing.

Another consideration for parts made with SLA, especially those with a long desired lifespan, is their aging characteristics. As resin parts are exposed to UV radiation over time, they continue to cure very slowly. This process causes them to become slightly stronger but noticeably yellower, more brittle, and less elastic over time.

Generative Design & Topology Optimization

Generative design: a computational design process that iteratively adjusts part geometry to meet design constraints and goals

Generative design uses FEA, CFD, and other numerical techniques to determine the best design in order to minimize mass, maximize strength, or meet certain thermal or flow requirements. The inputs are “preserved” and “obstacle” geometries that define how the part interacts with the rest of the assembly or the environment, as well as various loads, constraints, and goals. The software adds and subtracts material to connect the preserved geometries while avoiding the obstacles until the design goal is achieved or the iteration limit is reached. Topology optimization is a related process where the software is given a starting shape to iterate on instead of forming the initial iteration itself.

Both processes output unusual and organic geometries that can often only be manufactured with AM, making use for standard machined parts difficult. Despite this, generative design and topology optimization can help reduce weight and improve strength, addressing two important constraints of Battlebots applications.

The most common generative design software at the academic level is Autodesk Fusion 360. While perhaps better known for its CAD and CAM functions, Fusion 360 is relatively easy to use for generative design if you are familiar with Inventor or SolidWorks FEA and simulation. Fusion 360 allows the import of Inventor part and assembly files, as well as STEP and STL file types, among others.

→ Common Materials

Material Extrusion (FDM/FFF)

See the Materials Catalog for more materials and properties.

☆ These materials can be printed on the printers in the shop

Material	Filament Density (g/cc)*	Yield Strength (MPa)*	Characteristics & Use Case	Preferred Brand(s)
Polylactic acid (PLA)☆	1.24	40	Easy to print and nontoxic, general purpose material	Hatchbox, MatterHackers, ProtoPasta
Acrylonitrile butadiene styrene (ABS)	1.04	44	Better impact and heat resistance than PLA, general purpose material	Hatchbox, MatterHackers
Thermoplastic polyurethane (TPU)☆	1.22**	9**	Elastic and tough, good for armor and shock absorption	NinjaTek, Sainsmart, Overture
Nylon	1.14	50-60	Strong and very impact resistant, good for armor and structural parts at 3lb level	Ultimaker
Carbon fiber nylon	1.0	100	High rigidity and strength, good for structural parts	MatterHackers, Markforged

*These values may vary slightly by brand and formulation.

**Values are for NinjaTek Cheetah 95A, properties vary widely with hardness

Vat Photopolymerization (SLA)

Due to the more complex nature of the printing technology, quality SLA printers are not usually open-source, meaning they will only accept materials from a certain brand. The most common SLA printer manufacturer in general and on campus is Formlabs, and a few of the most commonly used and useful resins are listed below. Check the Formlabs website for more information about available materials.

Material	Ultimate Tensile Strength (MPa)	Notched IZOD Impact Resistance (J/m)	Characteristics & Use Case
Clear V5	60	29	Easy to print, good for detailed or complex parts and general use
Tough 2000	46	40	Sturdy and impact resistant, good for jigs and fixtures
Rigid 10K	65	47	Very strong, stiff, and impact resistant, good for most Battlebots uses where ductility is not important
Flexible 80A	8.9	Unknown	Elastic but not too soft, good for shock absorption and seals

Note: Listed material properties are post-UV cure values.

→ Slicing & Settings

Material Extrusion (FDM/FFF)

Slicing Software

UltiMaker Cura is one of the most popular amateur slicing softwares for FDM/FFF because it's easy to use, provides a reasonable amount of control over slicing settings, and can slice for many popular 3D printer brands. Prusa, the manufacturer of the shop printers, has a proprietary slicing software with similar functions. Eiger is the web-based slicer for Markforged printers. Bambu Studio is the slicing software released by Bambu Lab; the backend functionality is based on PrusaSlicer.

Software	Compatible Printers	Part File Types
UltiMaker Cura	All Ultimaker printers, Prusa i3 MK3/MK3S, other hobbyist printers	STL, OBJ, 3MF, AMF
PrusaSlicer*	All Prusa printers, Ultimaker 2, other hobbyist printers	STL, OBJ, 3MF, AMF, STEP/STP
Eiger	Markforged printers	STL
Bambu Studio	All Bambu Lab printers, Prusa MK3S/MINI, other hobbyist printers	STL, OBJ, 3MF, AMF, STEP/STP

*Preferred slicer for shop printers (Prusa i3 MK3S+)

Important Settings

Layer height: Layer height refers to the thickness of each layer of printed material. It affects the precision of small features and print time. Most slicing softwares have set speed and quality options for different layer heights that optimize physical and visual properties for specific applications. For example, Cura has a Engineering print profile for Ultimaker printers with 0.1 and 0.15 mm layer height options tuned for end-use parts with better tolerances and accuracy.

Infill: Infill affects the overall mass of the part and its strength and stiffness. It refers to material on the inside of the part. There are two settings that dictate the infill: infill percentage and pattern. Infill percentage refers to the density of the infill material. The recommended infill percentage varies with application, but breaking force tends to level off around 80%. Non-elastomeric materials with very high infill percentages also have a tendency to warp. For non-structural internal parts, 10-30% infill is sufficient.

Suggested Infill Patterns:

- **Lightning:** extremely fast print time, inconsistent strength along Z axis, more of an internal support structure than an infill; consider using more than 2 walls
- **Adaptive cubic:** light parts and fast print time, but still stable
- **Line/zig zag:** fast print times, not very strong
- **Grid:** balance between strength and print speed
- **Gyroid:** strong and light, similar strength in all directions with moderately fast print time, recommended infill pattern for non-flexible Battlebots applications
- **Archimedian chords/cross/cross 3D:** allow elastic materials to flex, recommended infill patterns for flexible Battlebots applications; Archimedian chords should be printed so the XY plane is parallel to the compression direction

Wall/top/bottom thickness: Wall thickness affects the overall mass of the part and has more effect on the strength and stiffness than infill. Increasing the number of walls, floors, and ceilings will make the part more rigid and stronger. For elastic materials, increasing the wall thickness decreases flexibility.

Temperature: The temperature of the nozzle and build plate don't affect the material properties of the part itself, but they do impact the printing process. Heating the build plate to the recommended upper limit for the material usually guarantees proper adhesion; do not go higher than the upper limit because it could impact the accuracy of the part's dimensions. A nozzle temperature 10-15° lower than the maximum reduces stringing in materials like nylon or TPU without sacrificing consistency.

Support: Overhangs greater than 45° usually require some kind of material supporting them. On dual-extrusion printers like the Ultimaker S3 and S5, the support material may be different from the primary material and is usually water-soluble or dissolvable in another common solvent. On printers with one extruder like the Prusas in the shop, the support material is the same as the primary material but is designed to be easy to remove. Support can be manipulated to be easier to remove, to make the supported surfaces smoother, or to print faster by changing the overhang cutoff angle, support infill settings, and contact settings.

Adhesion: For parts with large, flat bottom surfaces, low infill percentages, or weird thermal properties, adhesion may be necessary to reduce the risk of warping. (**Note: this does NOT mean you should put glue stick on the print bed.**) Adhesion comes in two main forms: brim and raft. Both add easy-to-remove printed material outside of the part volume to change the contact between the part and the print bed.

A brim is printed around the outside of the first layer to increase the surface area and pull the bottom of the part back down if it tries to warp. A raft is a few layers of material slightly larger than the bottom of the part printed underneath it. Instead of the bottom of the part warping, the raft takes the brunt of the heat effects from the print bed.

Vat Photopolymerization (SLA)

Slicing Software

The Formlabs proprietary software is called PreForm. The free version is only compatible with Formlabs printers and certified resins. One option for open-source software for non-Formlabs SLA and MSLA printers is Chitubox, which functions similarly to PreForm but allows you to slice for various printer brands. PreForm takes STL and OBJ file types, as well as FORM files previously generated in the slicer.

Important Settings

Layer height: Layer height determines the precision of small features, print time, and the smoothness of shallow curves. Most Formlabs resins allow for a choice between 50 and 100 micrometer (10^{-6} m; aka micron) layers; some also allow 25 or 200 micron layers. PreForm also has an Adaptive layer height setting which changes the height of each layer to maximize both speed and precision. 100 micron layers are usually good enough for most Battlebots parts, but in cases where there are small holes or shallow curves, 25 or 50 micron or Adaptive layers may produce a better part.

Orientation: While it is not a true print setting, orientation is important for parts printed with SLA. A good rule of thumb for blocky parts is to position them so that any large, planar faces are 15-75° from horizontal, especially thin sections. Keep in mind that any faces that are facing the build plate will likely have support structures attached to them if they are less steep than about 45° from vertical. Parts smaller than about 1” are less likely to warp, so orientation should be determined based on the best support configuration.

Support: Standard SLA supports are thin and tree-like, so density and contact settings are the most important factors. The 100% support density automatically chosen by PreForm is often unnecessary and can trap uncured resin on the surface of the part, so 85-90% is usually sufficient for parts of average complexity. The touchpoint size refers to the size of the contact patch at the interface between a support branch and the part. A touchpoint size of 0.4-0.5 mm is sufficient to support most parts while making support removal easy. Support locations generated in PreForm can be edited and removed to reduce the risk of support structures interfering with the part’s functionality, though this increases the risk of undersupported parts.

→ Materials Catalog (FDM/FFF)

Material	Density (g/cc)	Yield Strength (MPa)	Glass Temp. (°C)*	Impact Toughness (kJ/m^2)	Elongation at Break (%)
General polylactic acid (PLA)	1.2-1.3	20-50	50-60	1-5	2-5
MatterHackers PRO Series PLA	1.24	60	55	~5	6.0
eSun PETG (polyethylene terephthalate glycol)	1.27	52.2	64	4.7	83
UltiMaker ABS	1.10	38.1	100.5	14	4.6
Prusament PC Blend (polycarbonate blend)	1.22	63	**113**	12	5.8
UltiMaker Nylon	1.14	63	55	34	210
NylonX (carbon fiber-filled nylon)	**1.00**	**100**	~50	60	~2
NylonG (fiberglass-filled nylon)	**1.00**	95	~50	**80**	~2
NinjaTek Cheetah 95A (TPU)	1.22	9	-24	19	**580**
Sainsmart Flex 95A TPU	1.21	16	<0	15-25	450
Kimya PEBAX (93A flexible material)	1.01	33	<0	**No break (Charpy)**	>550

*Heat deflection temperature is generally slightly lower than glass temperature. In cases where glass temperature is not listed, heat deflection temperature is used. Elastomers have glass transition temps well below room temperature (see NinjaTek Cheetah, Sainsmart Flex, and Kimya PEBAX), which contributes to their flexibility.

Notes: Properties vary depending on print settings and geometry, so this table should primarily be used for comparison. Not all properties are available for each material, so some values are estimates or from comparable properties. Bolded/highlighted values are the best (highest/lowest) for each property based on Battlebots design considerations.

➔ On-Campus 3D Printing Resources

In addition to the Prusas in the shop, there are makerspaces throughout campus that provide 3D printer access to students. Here are makerspaces that Battlebots teams and members have used before:
** Info may be outdated*

➔ **Flowers Invention Studio (2nd Floor MRDC)**

- **FDM/FFF:** Ultimaker S5, Bambu Labs X1E w/ AMS, Markforged Mark Two
- **SLA:** Form 4, Form 3L
- **Stock Materials:** PLA, PETG, ABS, PVA, carbon fiber nylon, Formlabs resins

➔ **Aero Maker Space (Weber Building I)***

- **FDM/FFF:** Prusa i3 MK3, Stratasys Fortus 250mc, Stratasys F170, Stratasys F370 CR, Makerbot Method X
- **SLA:** Form 2, Form 3
- **Other:** Stratasys PolyJet printer
- **Stock Materials:** PLA, PETG, ABS, ABS-CF, ASA, QSR dissolvable support, nylon, carbon fiber nylon, TPU, Formlabs resins

➔ **The Hive Makerspace (Van Leer Building Rotunda)***

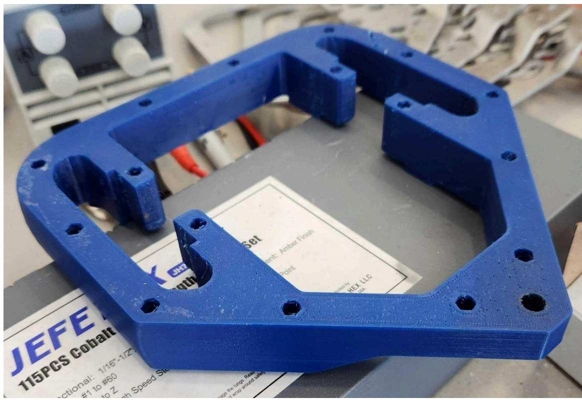
- **FDM/FFF:** Ultimaker S3, Stratasys F170, Markforged Mark Two
- **SLA:** Form 3, Form 3L
- **Stock Materials:** PLA, ABS, ASA, TPU, QSR dissolvable support, carbon fiber nylon, Formlabs resins

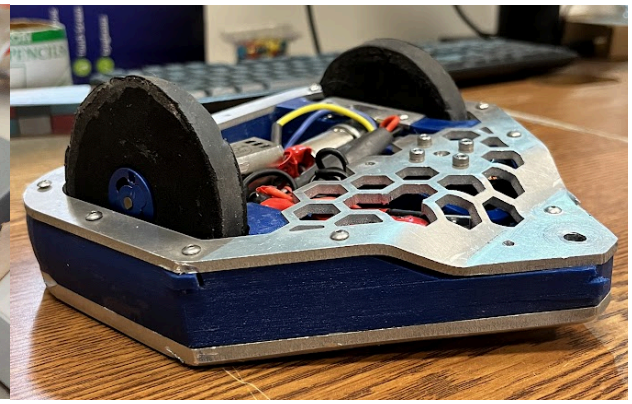
➔ **SEEDlab (GTRI employees only; Baker Building 020)**

- **FDM/FFF:** Ultimaker S5
- **SLA:** Form 3, Form 3B
- **Stock Materials:** PLA, ABS, PVA, nylon, Formlabs resins

→ Examples

TPU Chassis/Armor (Anomali, 3 lbs, 2022-23)





Notes

The chassis held up very well during the fights. There was minor delamination on one side, but it was not significant. However, these prints did take a significant amount of time to clean up due to old/water damaged TPU. In general, printing at the Aero makerspace takes about 4 days, and about half the time they forget to print your part.

Material & Settings

Printer: Prusa i3 MK3

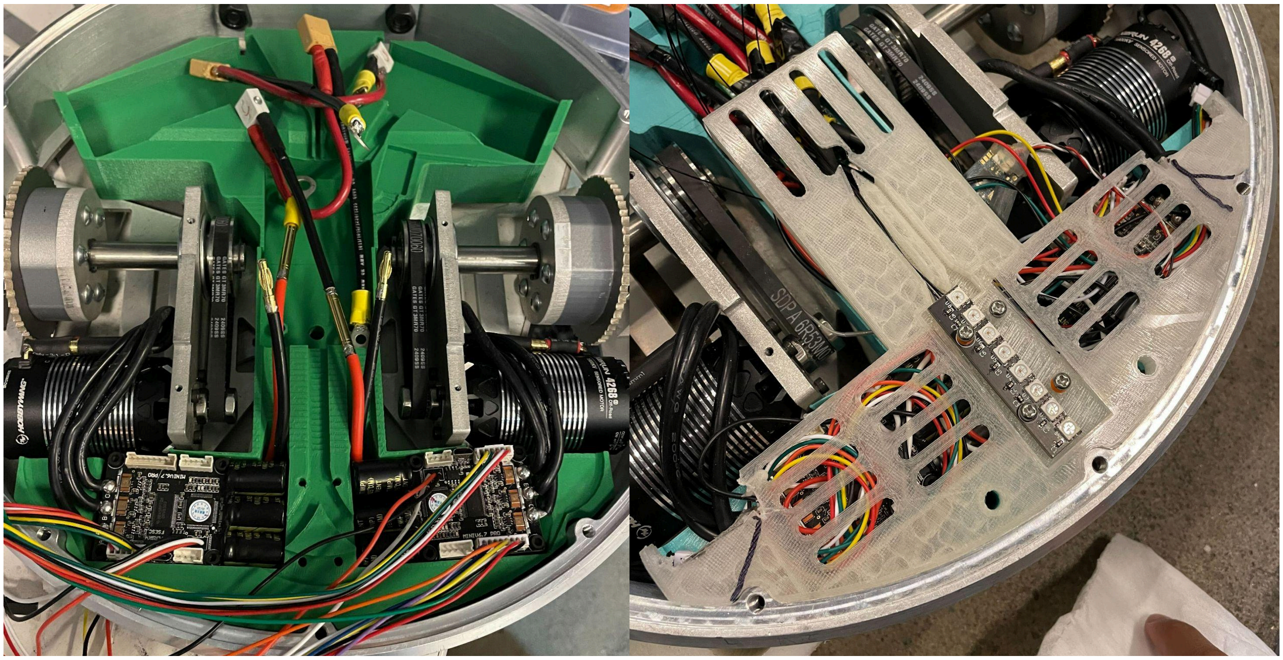
Material: Overture TPU

Layer Height: 0.15 mm

Infill: 15%, triangles

Wall thickness: 1.6 mm (4 walls)

Electronics Clamshell (Hockii, 12 lbs, 2022-23)



Notes

The purpose of the electronics clamshell was to hold the electronics in place to prevent wires from unplugging while in motion. It was held to the chassis by compression and bolts going through the PCB and bottom plate. The design went through around 10 iterations in two months, utilizing 3D printing's rapid prototyping and customization functionality.

Material & Settings

Printer: Ultimaker 3, S3, S5

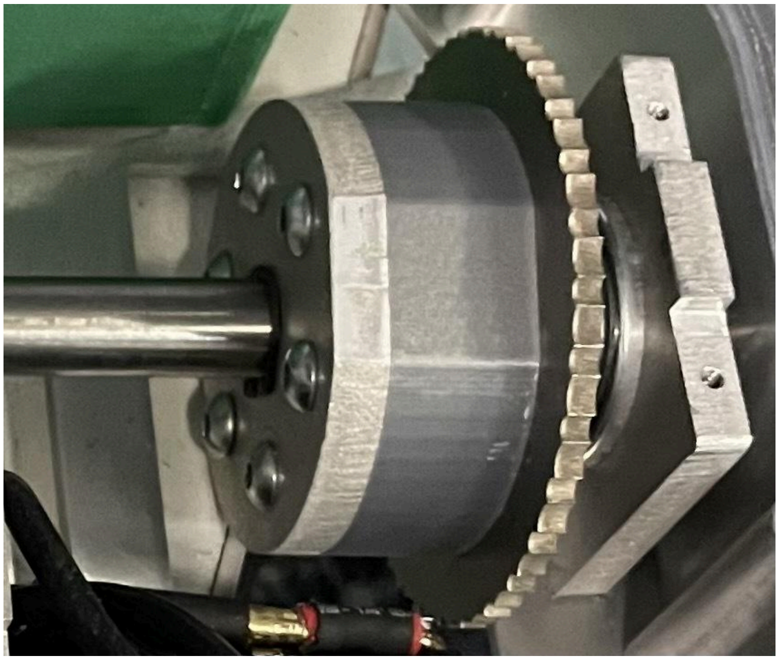
Material: PLA (various brands)

Layer Height: 0.2 mm

Infill: 10%, gyroid

Wall thickness: 1.2 mm (3 walls)

Custom Wheel Spacers (Hockii, 12 lbs, 2022-23)



Notes

The purpose of the wheel spacers was to add stability to Hockii's custom titanium cleat wheels while also making them easy to replace with Colson wheels by increasing the form factor. They interlocked with the titanium wheels and key to take a small amount of the shear load off of the bolts. The spacers made it easy to account for machining mistakes because each size could be a different width.

Material & Settings

Printer: Prusa i3 MKS+

Material: Hatchbox PLA

Layer Height: 0.2 mm

Infill: ~20%, gyroid

Wall thickness: 0.8 mm (2 walls)

Electronics Box (Papajonni, 3 lbs, 2021-22)

Notes

The purpose of the box was to contain and protect the electronics while also reducing weight. The original Onyx (carbon fiber nylon) chassis mostly survived Chonkii during the Motorama RJ rumble, but the thinner sections had cracked beyond further use. The decorated replacement nylon box was more impact resistant and flexible and had no notable weaknesses at the in-house competition.

Material & Settings

Printer: Markforged Mark Two, Ultimaker S5

Material: Markforged Onyx, Ultimaker Nylon (shown)

Layer Height: 0.2 mm

Infill: ~50%

Wall thickness: 0.8 mm (2 walls)

→ Tips & Tricks

Design & Slicing

- Tolerance for loose press fit in NinjaTek Cheetah (good starting point for other 95A TPU, too): ~2.5% (e.g., 0.256" for ¼" standoff)
- If you want to be able to drill out a hole in an FDM part, increase the wall thickness by 2-3 layers or more to guarantee the wall will not be breached
- When using the hole tool in Inventor or SolidWorks, set the fit to Loose for a normal fit and Normal for a close fit in FDM/FFF parts
- Using a raft when printing with nylon and nylon composites will reduce the risk of warping
- Cura has an Enable Ironing setting in the Top/Bottom menu in Custom Settings that noticeably smooths the top surface
- Enable Fuzzy Skin to increase the surface roughness of a part
- Adding small holes or pockets to FDM/FFF parts will increase the mass of the part; the effect decreases as infill increases
- If the ridge of layer seams is causing problems with your circular parts and features, set the seam position to Random

Materials & Finishes

- Nylon composites and other abrasive materials require a hardened steel nozzle for constant printing, which has different thermal properties than brass, so it may be necessary to tune the temperature when switching between nozzle materials
- NinjaTek Armadillo is known to have the tendency to shatter due to its high hardness

- It's difficult to completely eliminate stringing when printing TPU, but a heat gun can melt strings away and clear holes quickly, though be careful not to melt your part
- High-density TPU parts can be lightly heat-gunned to remove layer lines; doing this to low density (walls < 4) parts will probably make them cave in
- TPU and nylon are both hygroscopic, meaning they very readily absorb moisture from the environment; if parts start coming out stringier than normal, it is recommended to dry the filament in a low-temperature oven (instructions here) or a filament dryer for several hours (nylon should be printed from a filament dryer for best results)
- ESD resin is dissipative, meaning it is more conductive than an insulator; don't use it for electronics housings unless you know what you're doing
- Holes printed in Tough 1500 and 2000 resin can be tapped and reamed, though this should be done carefully by hand to avoid cracking
- ProtoPasta's HT-PLA can be heat-treated after printing to withstand temperatures up to ~140°C, though parts should be designed to compensate for later shrinkage