Xometry

Design Guide: Injection Molding

VERSION 2

Table of Contents

Overview

To get the most out of your Xometry Injection Molding experience, we offer the following part Design Guide with recommendations for part design, materials, surface finish, draft, wall thickness and undercuts. This Design Guide will help you achieve parts that are easily moldable while meeting your structural and cosmetic requirements. Your part may not conform to these guidelines and Xometry will gladly quote those parts, but it will take us a little bit longer to provide you with a quote.

Size Limitations

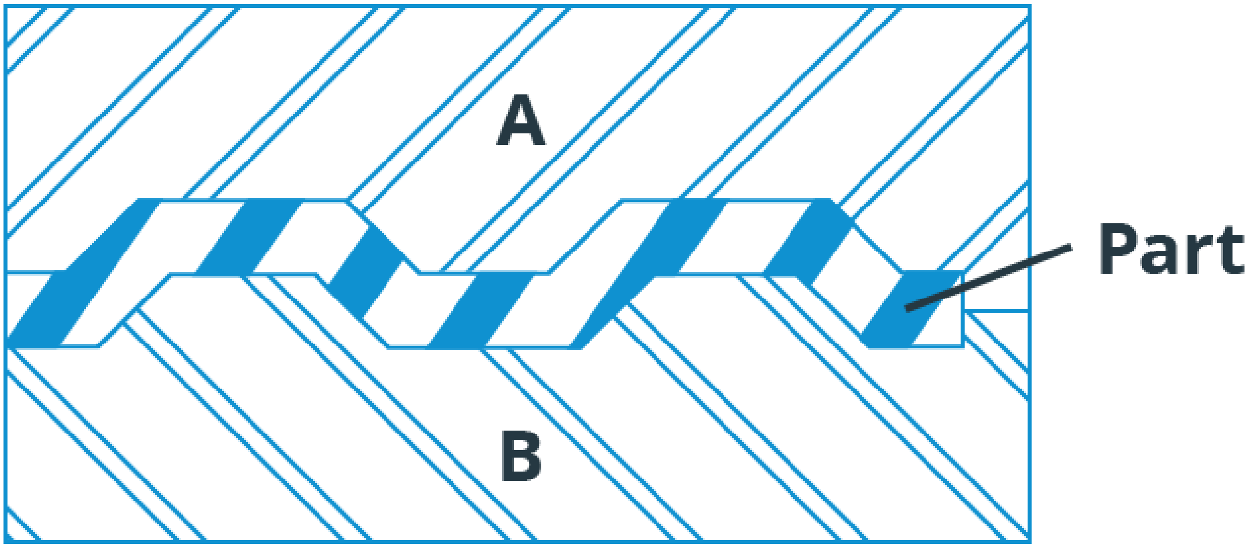
Injection molding can handle large parts as well as tiny parts. Due to Xometry's Manufacturing Partner Network, we can review mold opportunities of various shapes and sizes.

Plastic Part Tolerance

Injection molds are precision machined from aluminum or steel. These molds are typically machined to typical CNC tolerances of +/- 0.005", unless a tighter tolerance is specified. When plastic is injected into a mold, it cools and shrinks. The exact amount of shrinkage is a function of the resin being used. Each mold is machined slightly larger than the part to account for shrinkage of the resin when it cools. Although this shrinkage rate is very predictable, slight variations in the resin affect the shrinkage and hence the final part tolerance. The shrinkage variation gets larger as the part gets larger, so depending on the material you should expect the tolerance due to shrinkage to be roughly +/- 0.002 in./in. (0.05 mm/mm). The result is that a 4" ABS part will have a tolerance of roughly +/- 0.010-0.011" (.28 mm).

Part to part, the repeatability is very consistent, with little or no deviation.

Additional considerations that affect part tolerance include the registration of the two mold halves (in X and Y) and closure or seating of the two mold halves (in Z). These variables will affect features formed by the parting surfaces or shut-off surfaces (See illustration below). Fixed Dimensions are those dimensions not affected by mold closure and flash thickness variation. Closure Dimensions are those dimensions that can be affected by flash thickness variation.



If your tolerance needs are tighter than standard plastics tolerances, please let us know your design requirements and we will work with you to meet your needs.

Xometry's standards are as follows:

- Lead time stated is for first article shipment. Remaining production time is confirmed after first article approval.
- Typical first article shipments are 10 pieces.
- Xometry cannot guarantee a perfect color match per Pantone color.
- All quotes are based upon the assumption that designs have an adequate draft, radii, and coring for manufacturability. Design changes post-order may affect pricing and lead times.
- Cores, side actions, and tooling strategy are determined by Xometry unless explicitly discussed.
- Unless noted, tolerances are +/- 0.010" between the manufactured part and 3D CAD. Part-to-part repeatability is typically under 0.004".
- Gating, ejection, knit lines and parting lines are at the discretion of Xometry unless explicitly discussed.

Example of a gate location in a finished part

Example of knit lines in a finished part

Materials

The injection molding process offers a vast array of plastic resins to work with. These materials have been developed to meet specific requirements for strength, high temperature and chemical resistance, abrasion resistance and low friction. Thermoplastic resins fall into two distinct categories, each with their own advantages and disadvantages.

Amorphous Thermoplastics: For clear or opaque parts and include polycarbonate, acrylic, PETG, ABS and Polysulfone.

- **Advantages**: Bonds well using adhesives, very stable dimensionally and good impact resistance.
- **Disadvantages**: Low resistance to fatigue and stress cracking.

Semi-Crystalline Thermoplastics: These include the polyethylene family (LDPE, HDPE, UHMW-PE), Polypropylene, nylon, acetal and fluoropolymers.

- **Advantages**: Excellent for bearing, wear and structural applications, good chemical and electrical resistance' lower coefficient of friction.
- **Disadvantages**: Difficult to bond with adhesives, only average impact resistance.

These resins can be categorized as a) low cost commodity resins, b) medium cost engineering resins and c) high cost high performance/specialty resins (see table below).

AMORPHOUS RESINS	SEMICRYSTALLINE RESINS	COST
High-Performance/Specialty *Polyetherimide or PEI* Strength: High Heat & Chemical Resistance: High Applications: Aerospace	**High-Performance/Specialty** *Polyetheretherketone or PEEK* Strength: High Heat & Chemical Resistance: High Applications: Bearings, medical implants	**Expensive**
Engineering *Polycarbonate or PC* Transparent Strength: Moderate Heat Resistance: High Electric Insulator: High Applications: Electrical, windows	**Engineering** *Polymide or PA (Nylon)* Strength: Moderate to high Chemical Resistance: High Abrasion Resistance: High Shrinkage and Warp: Low Applications: Auto parts, textiles	**Moderate**
Commodity *Polystyrene or PS* Transparent Strength: Low Heat Resistance: Low Application: Cutlery, cups	**Commodity** *Polypropylene or PP* Flexibility & Toughness: High Chemical Resistance: High Fatigue Resistance: High Applications: Bottles, crates & cases, living hinges	**Inexpensive**

Xometry Injection Molding offers a wide array of resins for most applications. For the fastest quote response, select from the following list of materials. If the material you are interested in using is not on this list, If you plan to supply your own resin, please indicate your preference in the “Additional Information” section and send us a link to the MSDS and process data sheet documents.

MATERIAL NAME	ALTERNATE NAME
ABS	Acrylonitrile Butadiene Styrene
PC-ABS	Polycarbonate Acrylonitrile Butadiene Styrene, PC/ABS
Acrylic (PMMA)	Polymethyl Methacrylate
Delrin (Acetal, POM)	Polyoxymethylene
HDPE	High Density Polyethylene
LDPE	Low Density Polyethylene
Nylon 6/6	Nylon 66
Overmold	Multiple polymers or elastomers
PBT	Valox, Polybutylene Terepthalate
PC-PBT	Xenoy
PEEK	Polyether Ether Ketone
PEI (Ultem)	Polyetherimide
PET	Rynite, Polyethylene Terepthalate
PLA	Polylactic Acid
Polypropylene (PP)	Polypropylene
Polystyrene (PS)	Polystyrene
PPE-PS	Noryl, Polystyrene Polyphenyl Ethers PS/PPE
PPS	Polyphenylene Sulfide
PSU	Udel, Polysulfone
PVC	Polyvinyl Chloride
TPE (elastomer)	Santoprene, thermoplastic elastomer
TPV (rubber)	Rubber, thermoplastic elastomer vulcanized

Fillers & Additives

Part strength can be improved by the addition of reinforcing materials such as glass fibers. Fillers such as glass fibers can be added to many resins to make them stronger and stiffer, while at the same time making them slightly more brittle. Other fillers such as talc can be used to increase the hardness of molded parts. Other fillers such as glass beads and fused silica can be used to reduce the flexibility of a part, reduce warpage or minimize shrinkage. Additives such as UV stabilizers, static dissipating agent, or flame retardant can be added to the resin compound if required.

Note that fillers often change the look of the part, including a “splay” appearance showing dominant flow pathing and blotches from material fill.

A part made from a material with added filler

Surface Finishes

All molds are machined from aluminum and/or steel and the molded part will pick up even the smallest imperfection in the mold surfaces. The machining marks left by the end mills will be transferred to the molded part if not removed by bead blasting or polishing. In many cases, particularly on the inside of a part which is not visible to the eye, the tooling marks can be left on the mold and it will make no difference to the function or cosmetics of the part. Those surfaces which are visible, however, require additional finishing. Almost all molds are finished to some degree, but the more polishing required, the higher the cost of the mold tooling and the longer it will take to complete the mold.

SPI FINISH	DESCRIPTION	TYPICAL APPLICATIONS	ADDED EXPENSE
A-1	Grade #3, 6000 Grit Diamond Buff	High polish parts	$$$$$
A-2	Grade #6, 3000 Grit Diamond Buff	High polish parts	$$$$
A-3	Grade #15, 1200 Grit Diamond Buff	High-low polish parts	$$$$
B-1	600 Grit Paper	Medium polish parts	$$
B-2	400 Grit Paper	Medium polish parts	$$
B-3	320 Grit Paper	Medium-low polish parts	$$
C-1	600 Stone	Low polish parts	$
C-2	400 Stone	Low polish parts	$
C-3	320 Stone	Low polish parts	$
D-1	Dry Blast Glass Bead	Satin finish	$
D-2	Dry Blast #240 Oxide	Dull finish	$
D-3	Dry Blast #24 Oxide	Dull finish	$
MT110XX	A variety of matte surface textures that are more coarse than SPI D-3. Please specify which texture is required before submitting the project	Textured finish	$$$$

Continued on next page »

SPI FINISH	DESCRIPTION	TYPICAL APPLICATIONS	ADDED EXPENSE
MT112XX	A variety of almost leather-like texture options. Common for automotive interiors. Please specify which texture is required before submitting the project.	Textured finish	$$$$$
MT114XX	A variety of textures from lines, to swirls, to wood grain. Please specify which texture is required before submitting the project.	Textured finish	$$$$$
MT116XX	Pattern-style textures such as diamond, circular, or square grids. Please specify which texture is required before submitting the project.	Textured finish	$$$$$
Other Mold Texturing	Non MoldTech finishes or custom texturing	Textured finish	$$$$$+
As-Machined	No additional finishes are required.		$

An example of a part with an “as-machined” finish

Draft Angles on Part Surfaces

Unlike a CNC machined part which can have vertical walls, injection molded parts almost always have draft, or slightly slanted walls. This is because parts with vertical faces would become stuck in the mold, particularly on the b-side or core side of the mold as the part cools and contracts onto the core. Trying to push the part out of the mold would require too much force and risk bending the ejector pins.

Adding draft angles to the part surfaces are needed to facilitate ejection of the part from the mold. Most CAD systems allow you to add draft quite easily, but this should be done at the last stages of part design to keep complexity to a minimum. Since adding draft changes the overall shape of a part, it is not easy for your molder to know how to add draft and still preserve the intent of the part. Hence, in most circumstances, parts should be properly drafted when sent to the molder.

Not all surfaces require the same draft. Surfaces that are textured, for example, need more draft to avoid getting hung up in the mold. Here is a sample of the more common draft angles and their applications:

SURFACE DESCRIPTION	MINIMUM DRAFT
For “near-vertical” requirements	0.5°
Most common situations	2°
All shutoff surfaces	3°
Faces with light textures	3°
Faces with medium textures	5°+

Undrafted *Drafted*

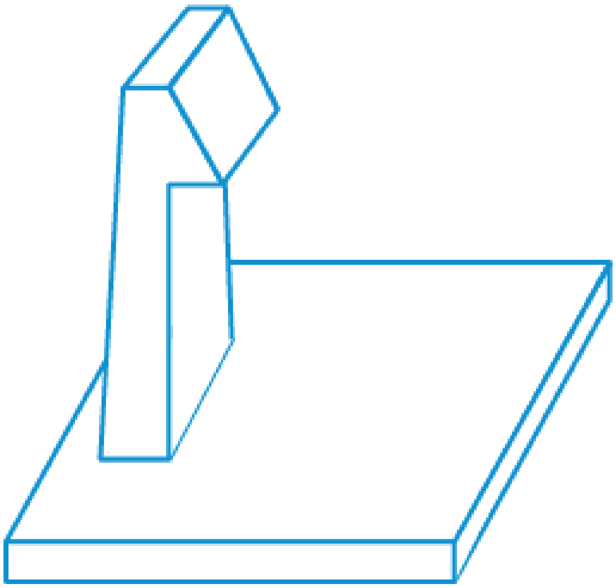
Undercuts

Many plastic parts can be molded in a straightforward two-part mold. More complex part geometries, and parts that contain snap fittings, have undercuts which prevent the part from ejecting from the mold. These undercuts are common and are created using a mechanism called a slide, side core, cam or hand load. If the undercut is on the outside of the part the side cores slide in from the side with each opening and closing of the mold. If the undercuts are on the same side of the part, more than one undercut can be handled by a single side core. Undercuts that are on the inside of a part are handled by a similar sliding mechanism called a lifter. Due to the constraints of the mold tooling and molding press, these side cores and lifters have a limited depth or pull.

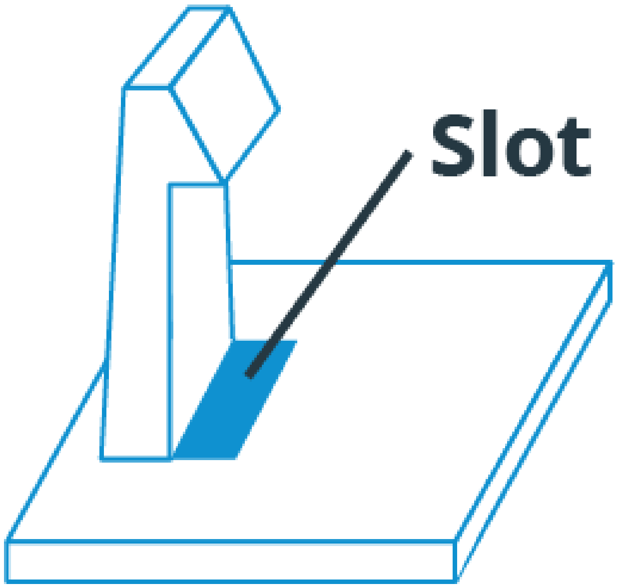
In many cases, the function of the undercut can be achieved by clever design without requiring a side core. If at all possible, use these design tricks to avoid the cost of a side core which can heavily affect tooling cost.

Snap Fits

While there are many reputable guides that go into depth about Snap Fits, one area that is often neglected is the importance of eliminating the slide or side core with proper design. In the case of a simple cantilever snap, the slide can be eliminated by providing a small slot at the base of the cantilever or by moving the snaps to the outside of the part. This is called a pass-thru core. You can see an example of this in the part shown.



Requires slide in mold



No slide required

Wall Thickness

Unlike CNC machined parts, plastic injection molded parts benefit from a consistent wall thickness. If a part is thicker in one section than another, a sink mark will appear at that location, possibly impacting the aesthetics of the part. Hence, a significant amount of design time for a properly designed part is spent “coring out” thicker sections and carefully designing ribs and bosses to ensure a consistent wall thickness throughout. Modern CAD design software has aided this task considerably with functions that allow the designer to “shell” a part to a specified thickness after the basic shape has been achieved.

Sink marks in finished parts

Selecting the proper wall thickness is a function of the selected resin (see chart below), but for many applications, a wall thickness of .060” - .100” is sufficient. If more strength or rigidity is required, rather than add wall thickness which increases the cost of the part, add a set of evenly distributed ribs.

Thin Walls Decrease Cycle Time

Cycle time equals cost so reducing cycle time is an important design goal. Parts with thick walls take longer to cool, increasing cycle time. Hence, the thinner you can design the walls of your part, the shorter the cycle time and the lower the cost of your part. In theory, cooling time is proportional to the square of the heaviest part wall thickness.

Warpage

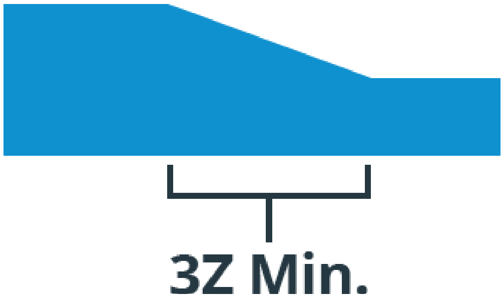
All parts shrink as the resin cools in the mold. Parts with uniform wall thickness shrink uniformly and avoid warp and sink marks. Non-uniform walls will cool and shrink at different rates, increasing the amount of warp inherent in the part design. Whenever our molding experts spot a potential problem due to non-uniform wall thickness, we will alert you and recommend that you modify your part using a uniform wall thickness.

If it is necessary to have a non-uniform wall thickness, the change in thickness should not exceed 15% of the nominal wall thickness and always use a smooth or tapered transition to ensure a successfully molded part.

MATERIAL	RECOMMENDED WALL THICKNESS
ABS	0.045 in - 0.140 in
Acetal	0.030 in - 0.120 in
Acrylic	0.025 in - 0.500 in
Liquid Crystal Polymer	0.030 in - 0.120 in
Long-Fiber Reinforced Plastics	0.075 in - 1.100 in
Nylon	0.030 in - 0.115 in
Polycarbonate	0.040 in - 0.150 in
Polyester	0.025 in - 0.125 in
Polyethylene	0.030 in - 0.200 in
Polyphenylene Sulfide	0.020 in - 0.180 in
Polypropylene	0.035 in - 0.150 in
Polystyrene	0.035 in - 0.150 in
Polyurethane	0.080 in - 0.750 in

Not recommended

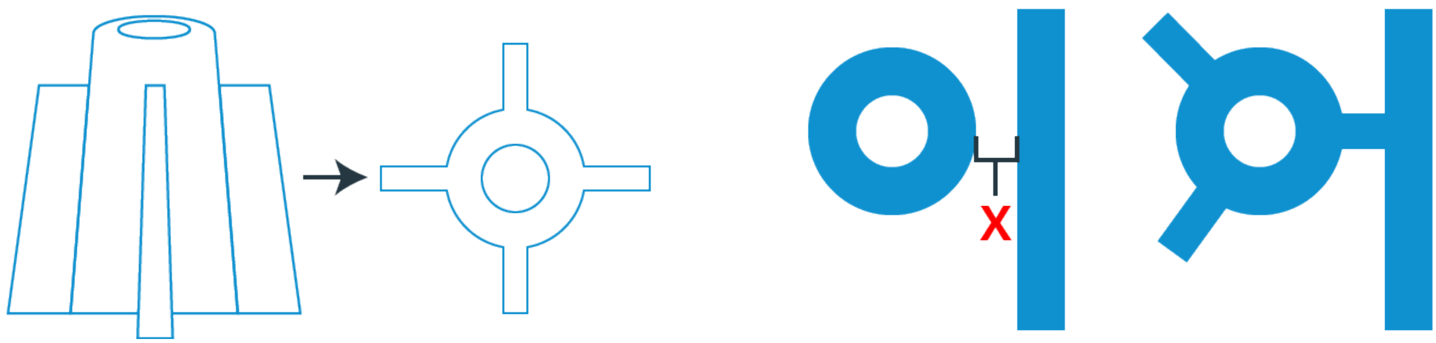
Recommended



Bosses, Ribs, & Gussets

Bosses are cylindrical standoffs molded into a plastic part to accept an insert or a self-tapping screw or pin for assembling or mounting parts. Bosses should be attached to a side wall or rib to enhance the structural rigidity of the part. The outer diameter (OD) of the boss should be 2 ½ times the diameter of the screw diameter for self-tapping applications.

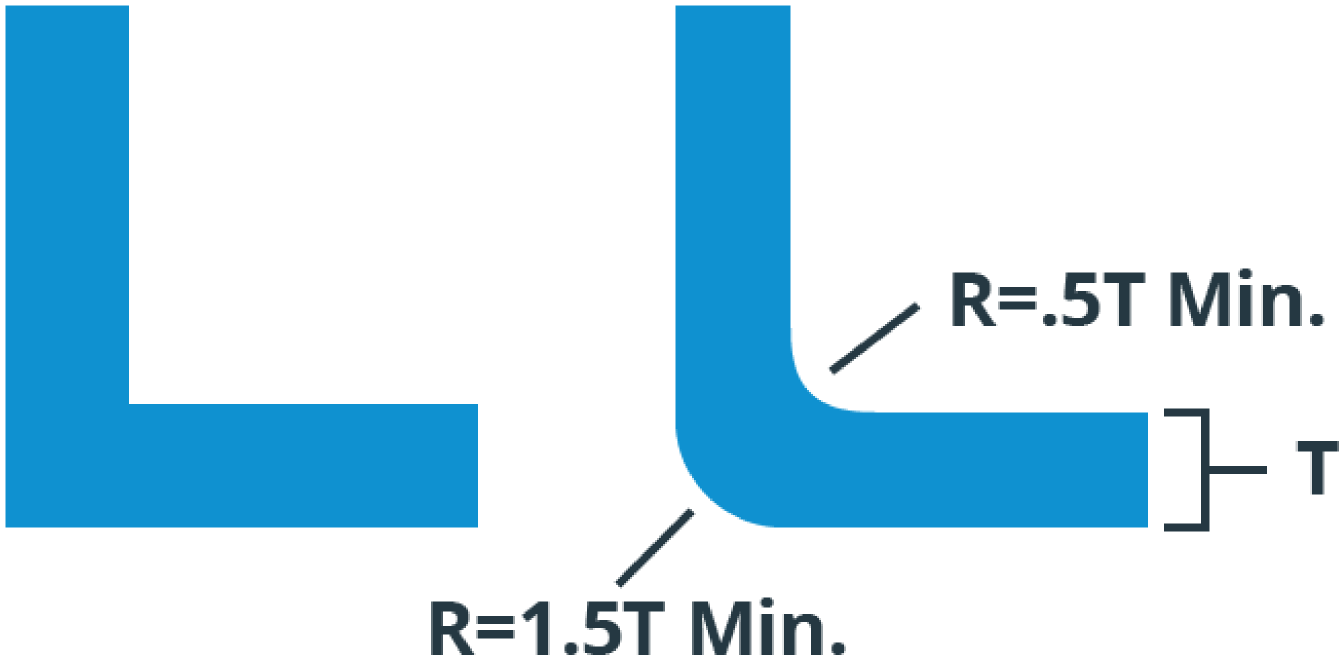
Where extra strength is required, many designers will add internal ribs to stiffen flat walls or support external loads. Similarly, gussets are ribs which connect a boss to the floor of a part. The thickness of a rib or gusset should be no more than 60% of the overall part thickness to minimize visible sink marks on the outside of the part. For example, a part with an outer wall of 0.120" should have internal ribs at about 0.070" thickness.



Bosses should be attached to a side wall or rib rather than freestanding

Part Radii & Fillets

As previously discussed, all molds are machined out of either aluminum or steel using machine tools and rotating end mills. As a result, certain geometries, such as sharp internal corners, are difficult to create without reverting to more expensive manufacturing techniques such as electronic discharge machining or EDM. To keep the cost of the mold as low as possible, eliminate all sharp corners that might will require the use of EDM. In addition, sharp corners within the mold can weaken the resulting part as molten plastic is forced to flow across or fill in a sharp corner. For that reason, use your CAD system's filleting tool to add a fillet and create smooth transitions between the walls and features of your part. The only place on your part where sharp corners are naturally created is at the parting surface or other shut off surfaces.



Not recommended

Recommended

Resources at Xometry

Online Instant Quoting

Web: Upload your CAD file at get.xometry.com/quote

CAD: Download the free Xometry Instant Quoting Engine℠ Add-Ins for SOLIDWORKS and Inventor: xometry.com/cad

Accepted File Types: .step, .stp, .sldprt, .stl, .x_t, .x_b, .ipt, .3dxml, .catpart, .prt, and .sat (max file size: 100MB)

Capabilities: Get an instant quote for CNC Machining, Sheet Metal Fabrication, 3D Printing (DMLS, FDM, PolyJet, SLA, SLS), and Urethane Casting. Request a manual quote for Injection Molding.

Live Engineering Support

Hours: M-F 8:00 AM - 9:00 PM EST

Email: support@xometry.com

Phone: (240) 252-1138

Online: support.xometry.com offers live chat, FAQs, and other helpful articles.

ITAR Registered

All uploads are secure and confidential. Contact us for assistance with large orders and existing Xometry quotes.

Need to use a PO? We offer same day credit application processing. Contact us for details.