
WEBINAR:
Injection Molding
with Xometry

Xometry

Your One-Stop Shop for
Manufacturing On Demand

Webinar
December 12, 2017

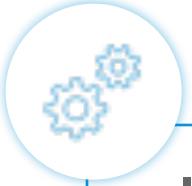
Greg Paulsen
Director of Applications Engineering
gpaulsen@xometry.com

Travis Minyard
Cost Engineer - Injection Molding
tminyard@xometry.com

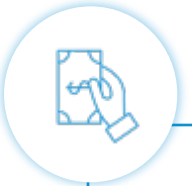
XOMETRY FOR ENGINEERING AND SOURCING PROFESSIONALS

A Simple Elegant Experience

Define Parameters



Instant Online Quote/RFQ



Buy

New Process: Injection Molding

Low Volume Production Manufacturing

INJECTION MOLDING BASICS

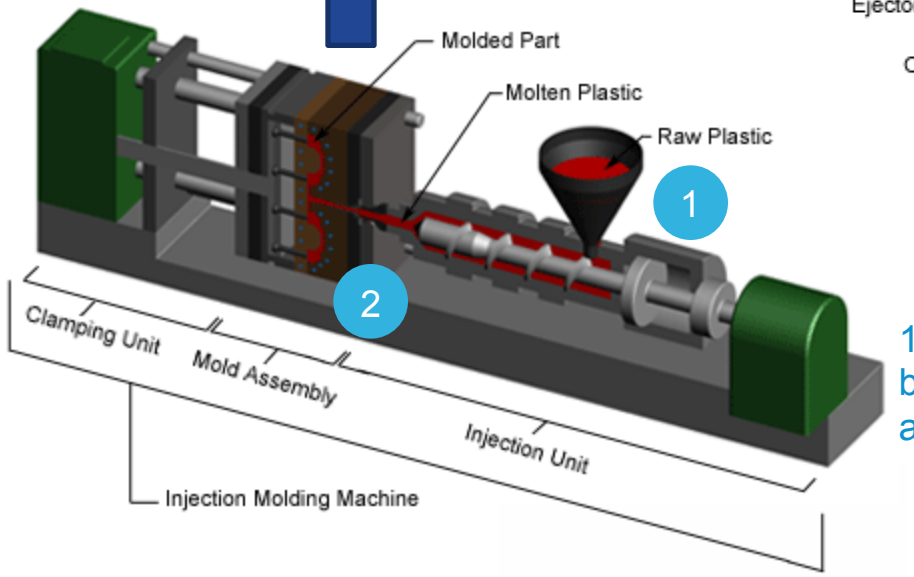
1. Injection Molding in a Nutshell
2. 5 Common Mistakes when Designing for Injection Molding
3. The Top Cost Drivers for Injection Molded Parts
4. How to Get Your Parts Molded at Xometry
5. Q & A

The Injection Molding Process

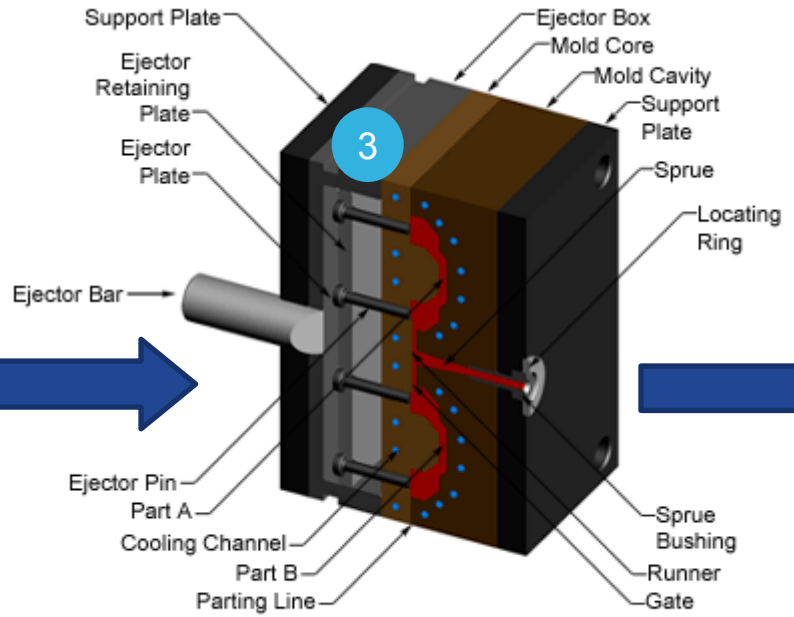
Travis Minyard
Cost Engineer- Injection Molding
tminyard@xometry.com

The Process

2) The screw inside the barrel moves the material towards the front of the barrel. A check valve forces the material into the mold at high pressure

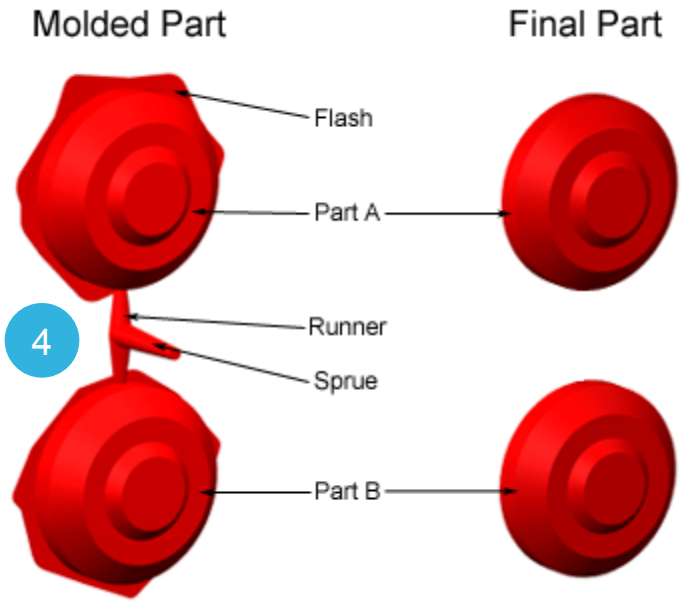


1) Material enters the barrel of the machine and begins to melt.

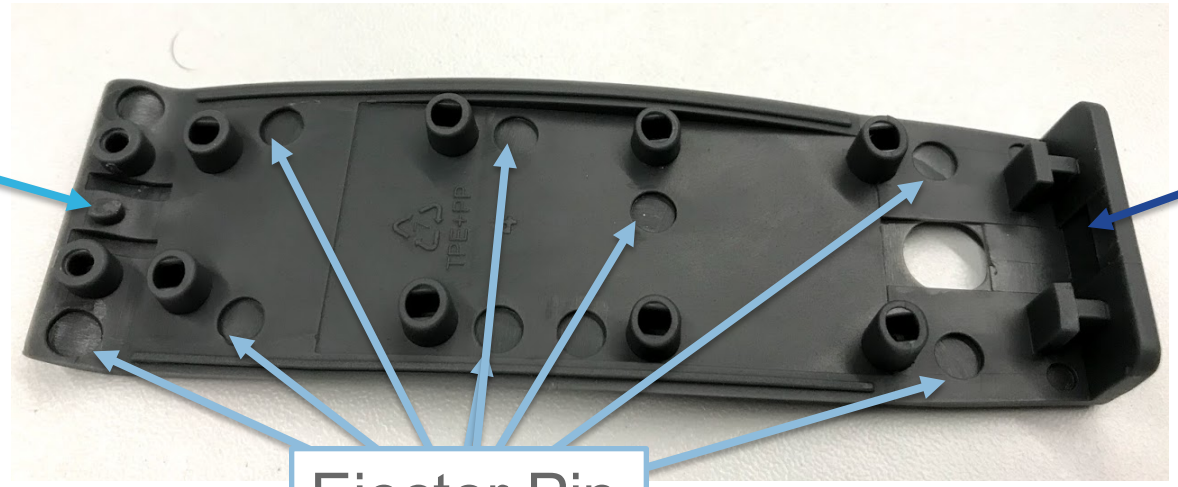


3) Part is cooled, mold opens and part ejects.

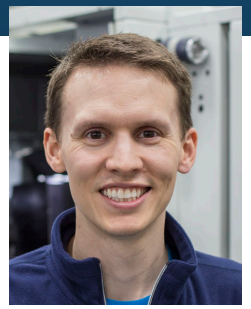
4) Runner and sprue are removed leaving a final part



A Typical Part (TPE, flexible)



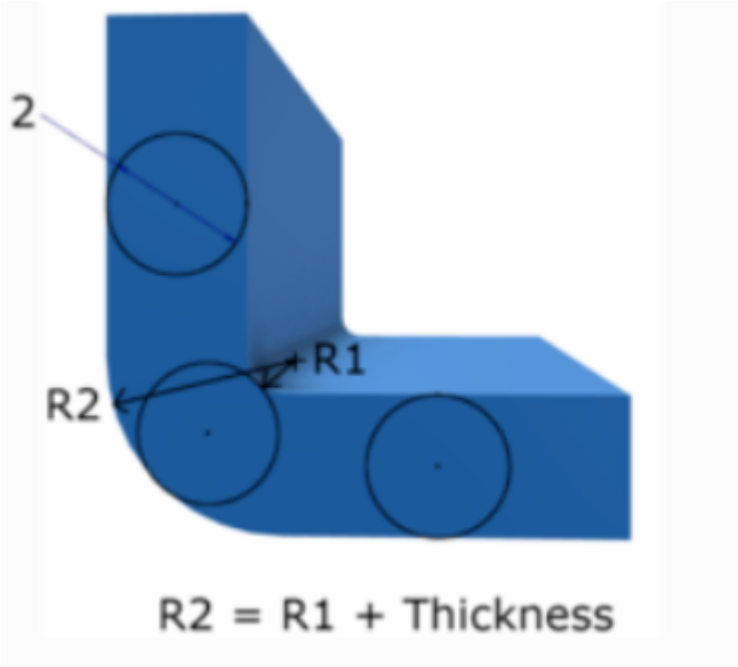
5 Common Mistakes when Designing for IM



Greg Paulsen
Director of Applications Engineering
gpaulsen@xometry.com

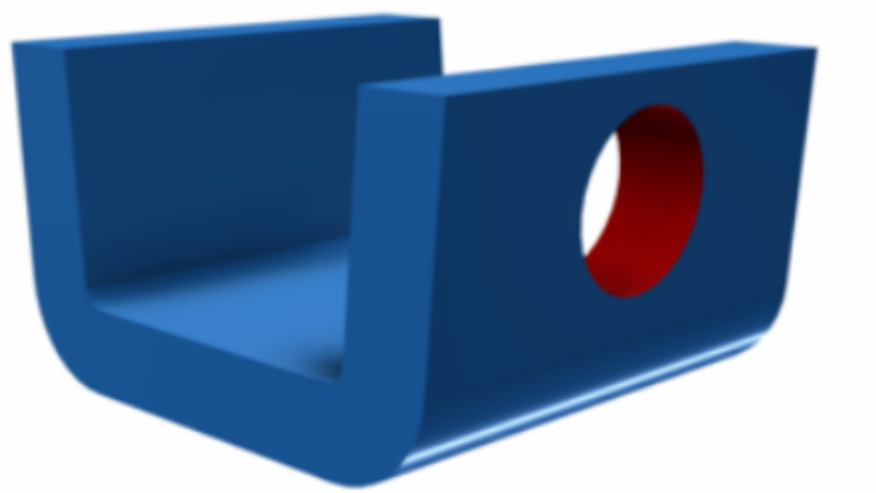
Not adding draft

Draft is needed in the mold to release the plastic part from the mold. Plastic generally shrinks towards the center of the part. Without draft, the part will stick in the mold and will not eject correctly. Design in 1° per side if possible, but any draft will be helpful to release the part.



Uneven wall thicknesses

Varying Wall Sections are common in part designs and can cause unintended part properties when molding. Plastics have a hard time flowing and transitioning between uneven wall thicknesses. Thick wall sections create differential shrinkage which may lead to part deflection. Try to keep the walls at least thicker at the gate area and thinner at the end-of-fill. Shoot for .080-120" wall thickness and utilize ribs and gusset features in your part design.



Unnecessary undercuts

Undercuts are designed-in, and often overlooked due to the complex nature of part design today. An undercut in the mold will need to be released before the part ejects or it will tear the feature out. There are many ways to release the undercuts with thru-coring design, slides, cores or lifters in the mold action. Simple rule for tooling: Undercuts+complex mold= more money.

Choosing the wrong materials

Material Selection is another common mistake I see. Material should be chosen based on where the part will live. Parts that will live outside should have UV stabilizers in them to prevent cracking. Parts that are under load, should have a filler in them like fiberglass to strengthen the part. Parts that have a bearing surface should have an additive like a lubricant.

Not embracing radii

Corner Radius is often overlooked in the part design. Corner radii strengthens that area and makes it more robust in the long term. Radii reduces stress concentrations and fractures in the plastic part. It also makes the part look aesthetically pleasing and lowers any risk of injury when handling.

WHAT DRIVES COST ON INJECTION MOLDED PARTS?

Travis Minyard
Cost Engineer- Injection Molding
tminyard@xometry.com

WHAT DRIVES COST ON INJECTION MOLDED PARTS

1. Material Selection

- Where will the part live?
- Material properties
- Price per pound

- ABS Acrylonitrile butadiene styrene
- PA Nylon 6, 6/6 Polycaprolactam
- HDPE High Density Polyethylene
- LDPE Low Density Polyethylene
- PP Polypropylene
- PBT Polybutylene Terephthalate
- PC/ABS Polycarbonate/ Acrylonitrile butadiene styrene
- PC/ PBT Polycarbonate / Polybutylene Terephthalate
- PET Polyethylene terephthalate
- PLA Polylactic Acid
- PMMA/Acrylic Polymethyl methacrylate
- PS/PPE Polystyrene/ polyphenyl ethers
- PC Polycarbonate
- POM Polyoxymethylene
- PS Polystyrene
- PSU Polysulfone
- TPE/ TPV Thermoplastic Elastomer/ Vulcanized

2. Cycle time

- Material cooling time
- Mold cooling
- Mold build
- The quicker the cycle, the less overhead per part

3. Surface finish

- Understanding your part finish
- SPI finish cart
- Higher the luster, higher the price

Finishes

SPI B-1

400 Grit Paper. Typical Application: Medium polish parts.

SPI B-2

400 Grit Paper. Typical Application: Medium polish parts.

SPI B-3

320 Grit Paper. Typical Application: Medium- Low polish parts.

SPI C-1

600 Stone. Typical Application: Low polish parts.

SPI C-2

400 Stone. Typical Application: Low polish parts.

SPI C-3

320 Stone. Typical Application: Low polish parts.

SPI D-1

Dry Blast Glass Bead. Typical Application: Satin Finish

SPI D-2

Dry Blast #240 Oxide. Typical Application: Dull Finish

SPI D-3

Dry Blast #24 Oxide. Typical Application: Dull Finish

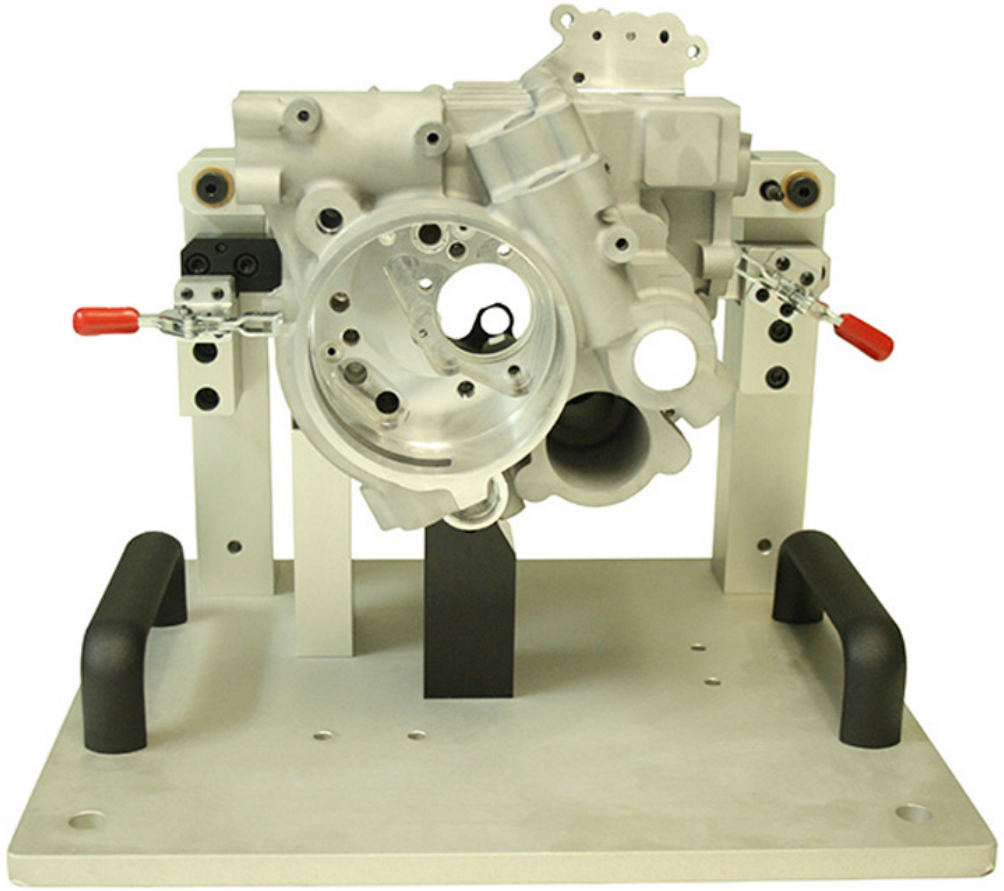
AS MACHINED

No secondary polishing or grinding. Part will show tooling marks.

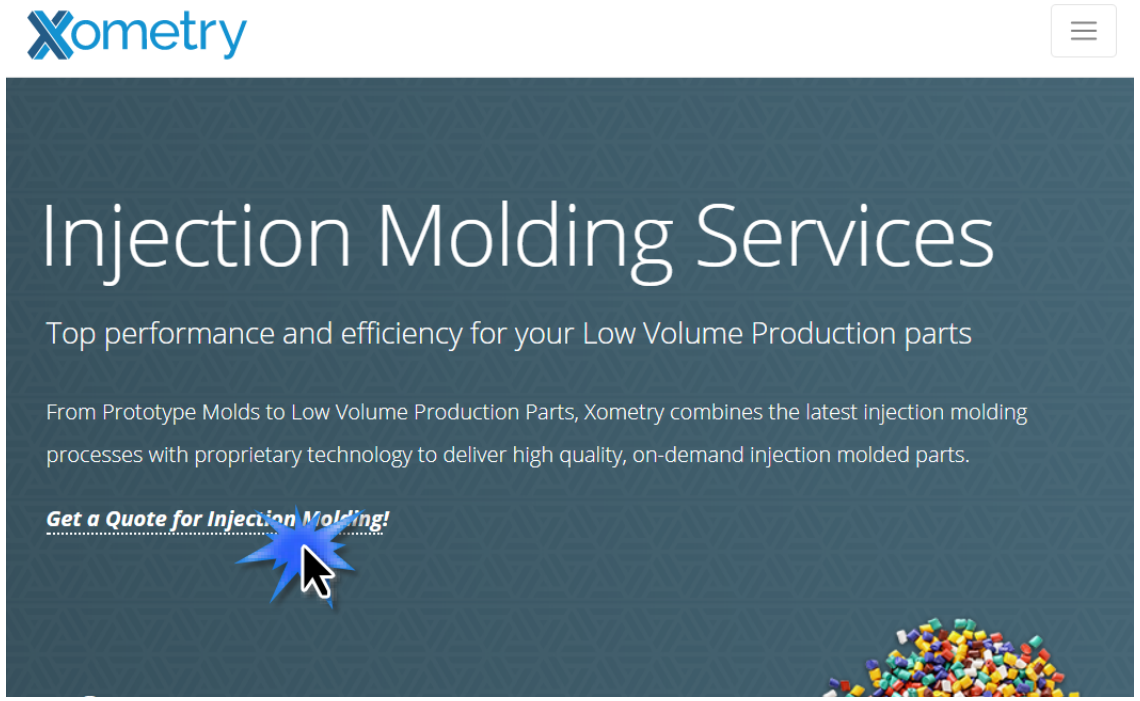
Matte finish

4. Post-molding Operations

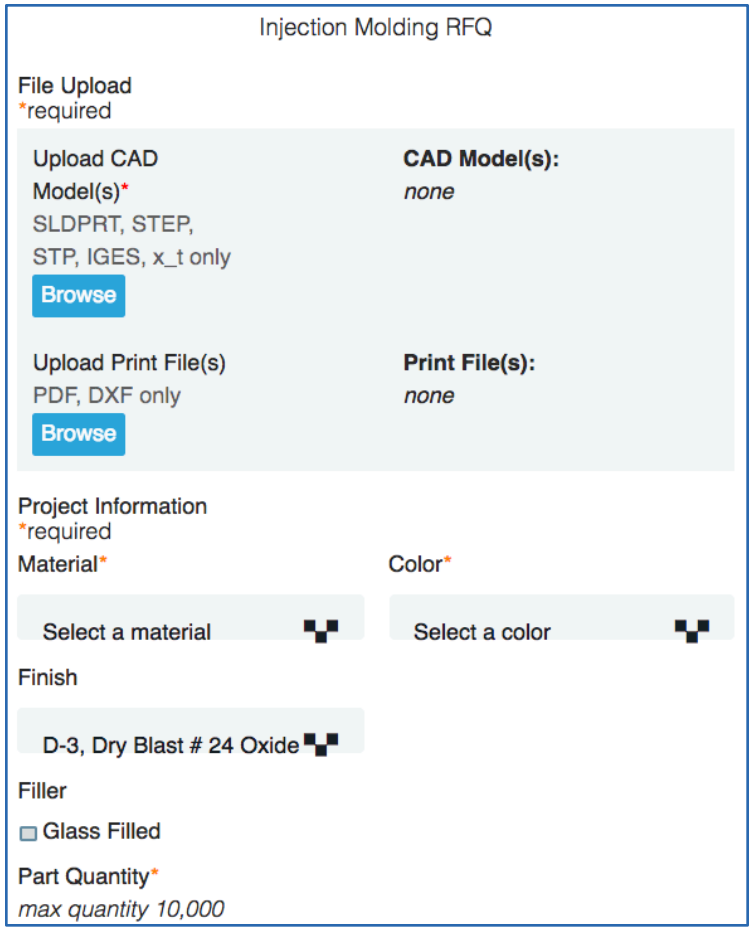
- Machining
- Cooling fixtures
- Packaging
- Special handling
- Special inspection
- Testing



XOMETRY'S INJECTION MOLDING SERVICES



https://www.xometry.com/injection-molding-service

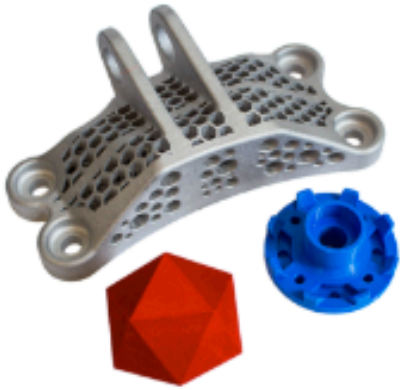


RFQ response within a business day!

XOMETRY'S OTHER SERVICES

CNC Machining

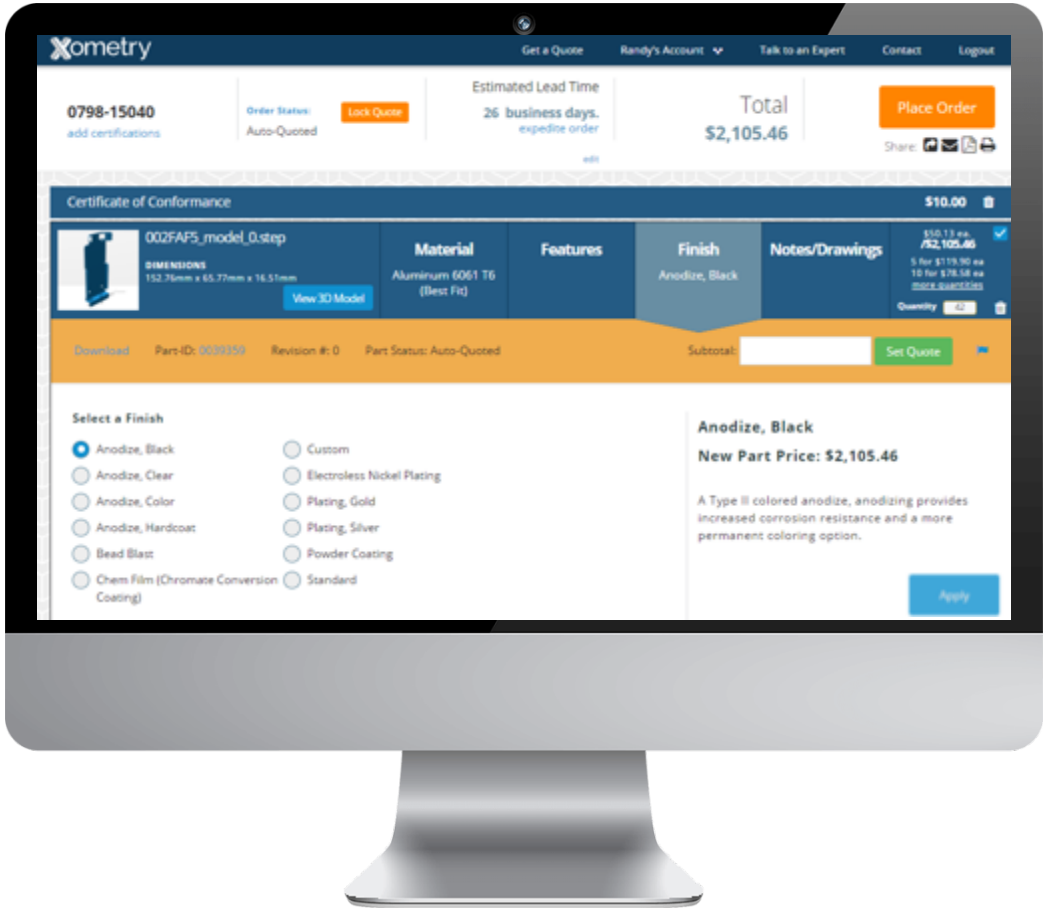
Sheet Metal



3D Printing

Urethane Casting

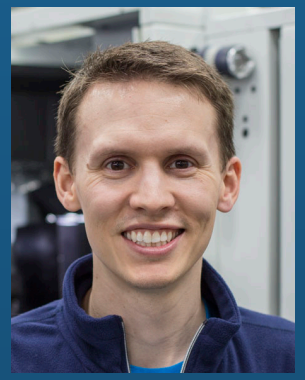
XOMETRY PLATFORM & SOLIDWORKS ADD-IN



- ✓Instant RFQ
- ✓Manufacturability Analysis
- ✓Dynamic Pricing & Lead Times

QUESTIONS?

Watch: xometry.com/youtube



Greg Paulsen
Director of Applications Engineering
gpaulsen@xometry.com

Travis Minyard
Cost Engineer- Injection Molding
tminyard@xometry.com

Xometry

Thanks for joining!

Support:

240-252-1138

support@xometry.com

Live chat on xometry.com