fictiv

INJECTION MOLDING DESIGN GUIDE

The Injection Molding Process

Injection molding is a fast and efficient way to produce plastic parts. The tool that it uses, an injection mold, has two main parts: the core and the cavity. These two halves of the mold come together, and molten plastic is injected into the tool.

Once the molten plastic cools, the two halves of the mold separate and the part is ejected. Depending on the construction of the mold and the part quantities that you need, this process can be repeated tens, hundreds, or many thousands of times.

That's just a high-level overview, of course, but it lays the groundwork for the details you'll find in this design guide. As you keep reading, remember that injection molding design is a three-part process.

1. Start your design
2. Upload your design and get design for manufacturing (DFM) feedback
3. Review part samples

This guide covers it all. **Let's get started.**

Step 1: Start Your Design

Step 1: Start Your Design

Starting your part design comes with ten major considerations.

- A Material Selection
- B Wall Thickness
- C Transitions
- D Corners
- E Draft
- F Ribs and Bosses
- G Tolerances
- H Parting Lines
- I Gates
- J Ejector Pins

Material Selection

Ⓐ Material Selection

Injection molding materials range from commonly used polymers to specialty plastics and polymer blends. There are literally hundreds of different plastic resins available, and they each have different end-use properties and processing requirements. Plus, there can be different grades of the same plastic material, including resin types with glass fibers or carbon fibers.

INJECTION MOLDED PLASTICS

Resin	Characteristics
Acetal (POM)	Excellent rigidity and thermal stability with low water absorption and good chemical resistance
Acrylic (PMMA)	Strong, lightweight, shatter-resistant, optically clear, and UV and weather-resistant
Acrylonitrile Butadiene Styrene (ABS)	Strong and impact-resistant, even at low tempera-tures
Nylon (PA)	Tough with high heat resistance, high abrasion resis-tance, and good fatigue resistance
Polybutylene Terephthalate (PBT)	Resistant to creep and used in parts with thin cross-sections
Polycarbonate (PC)	Strong, lightweight, and naturally transparent, with stable properties over a wide temperature range

INJECTION MOLDED PLASTICS

Resin	Characteristics
Polyether Ether Ketone (PEEK)	Excellent mechanical properties and resistance to chemicals and thermal degradation
Polyetherim-ide (PEI)	Combines stiffness and stability with low flammability and low smoke production
Polyethylene (PE)	Generally used for indoor applications. Chemically resistant. Includes high and low-density materials.
Polyphenyl-sulfone (PPSU	High toughness, high flexural and tensile strength, and good resistance to chemicals and heat
Polypropylene (PP)	Good chemical resistance and won’t degrade when exposed to moisture or water
Polystyrene (PS)	Lightweight, relatively inexpensive, and resistant to moisture and bacterial growth
Thermoplas-tic Elastomer (TPE)	Processed like plastic but has the properties and per-formance of rubber
Thermoplastic Polyurethane (TPU)	Rubber-like elasticity with good load-bearing capa-bilities

Wall Thickness

Ⓑ Wall Thickness

Wall thickness affects the strength, cost, and appearance of your injected molded part. Simply put, it's one of the most important design considerations.

There are three wall thickness terms that designers need to understand:

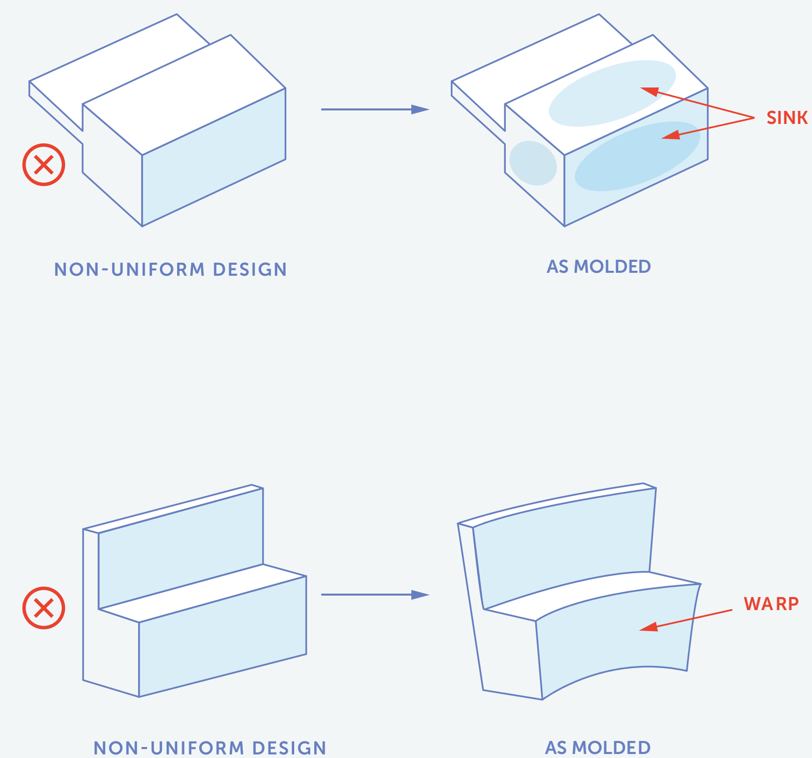
- Uniform wall thickness
- Nominal wall thickness
- Recommended wall thicknesses

UNIFORM WALL THICKNESS

Whenever possible, apply a uniform wall thickness to your part. This best practice promotes more consistent cooling and, consequently, more consistent shrinking.

You can use walls with different thicknesses,
but they'll cool and shrink at different rates.
This may cause sink, warp, and other injection molding defects.

NON-UNIFORM WALL THICKNESS
Can cause distortion when molded



NOMINAL WALL THICKNESS

Nominal wall thickness is the thickness throughout your part. A uniform wall thickness is recommended, but it's also important to avoid walls that are too thick or too thin.

- **Walls that are too thick** require more plastic material and longer machine cycle times, both of which add costs to your project.
- **Walls that are too thin** may trap air where the plastic doesn't fill the mold completely. These "short shots" result in incomplete parts.

In addition, keep wall thickness (or thicknesses) within range for the plastic material you've selected. The next section lists the recommended wall thicknesses for common plastic resins.

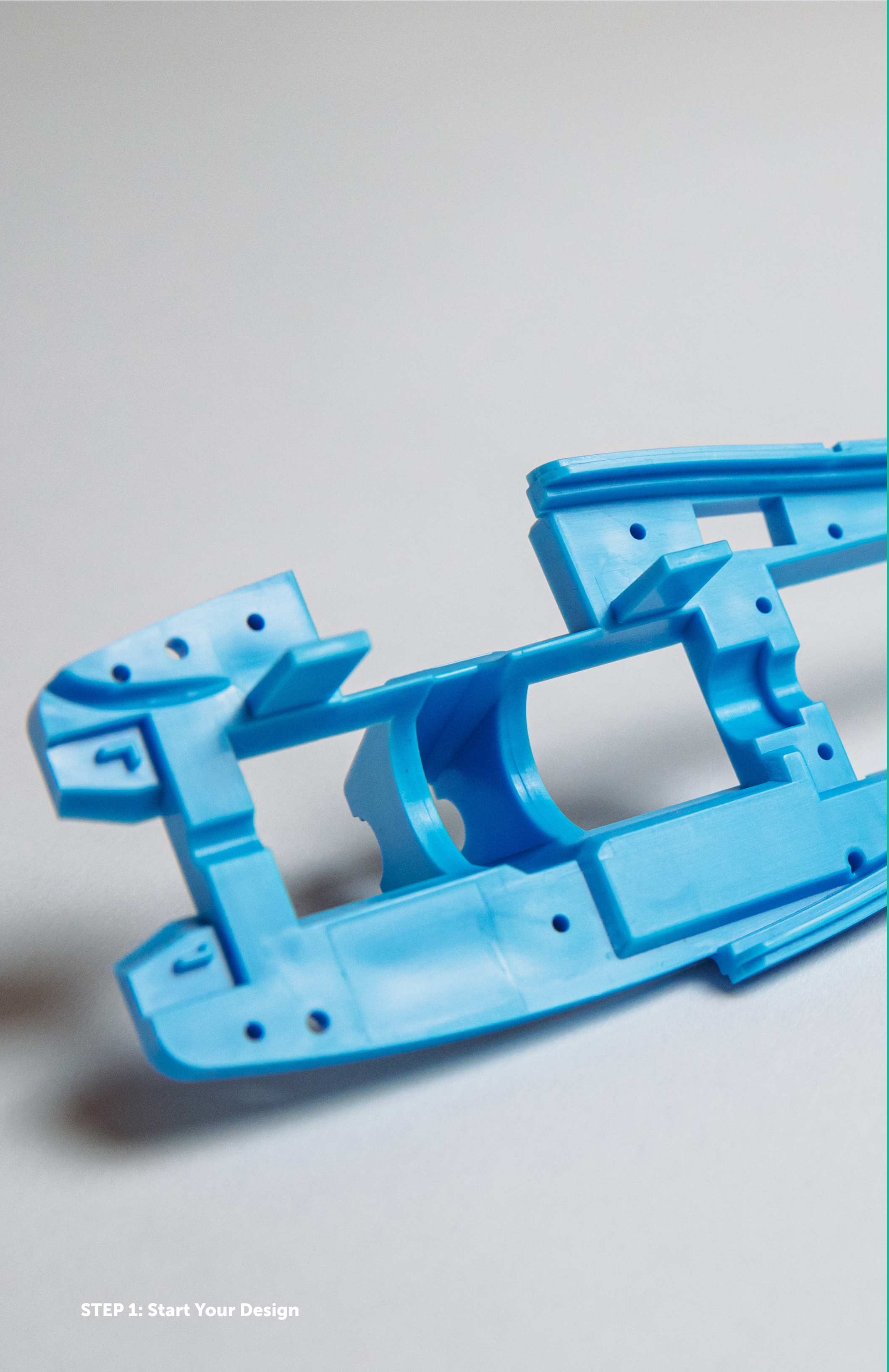
RECOMMENDED WALL THICKNESS

Resin	in	mm
Acetal (POM)	0.030–0.120	0.76–3.05
Acrylic (PMMA)	0.025–0.150	0.025–0.150
Acrylonitrile butadiene styrene (ABS)	0.045–0.140	1.14–3.56
Nylon (PA)	0.030–0.115	0.76–2.92
Polybutylene Terephthalate (PBT)	0.080-0.250	2.032-6.350

RECOMMENDED WALL THICKNESS

Resin	in	mm
Polycarbonate (PC)	0.040–0.150	1.02–3.81
Polyether Ether Ketone (PEEK)	0.020-0.200	0.508-5.080
Polyetherimide (PEI)	0.080-0.120	2.032-3.048
Polyethylene (PE)	0.030–0.200	0.76–5.08
Polyphenylsul-phone (PPSU)	0.030-0.250	0.762-6.350
Polypropylene (PP)	0.040–0.150	1.02–3.81
Polystyrene (PS)	0.025–0.125	0.64–3.18
Thermoplas-tic Elastomer (TPE)	0.025–0.125	0.64–3.18
Thermoplastic Polyurethane (TPU)	0.025–0.125	0.64–3.18

If you can't maintain a uniform wall thickness throughout your design, use smooth transitions between sections with different thicknesses.



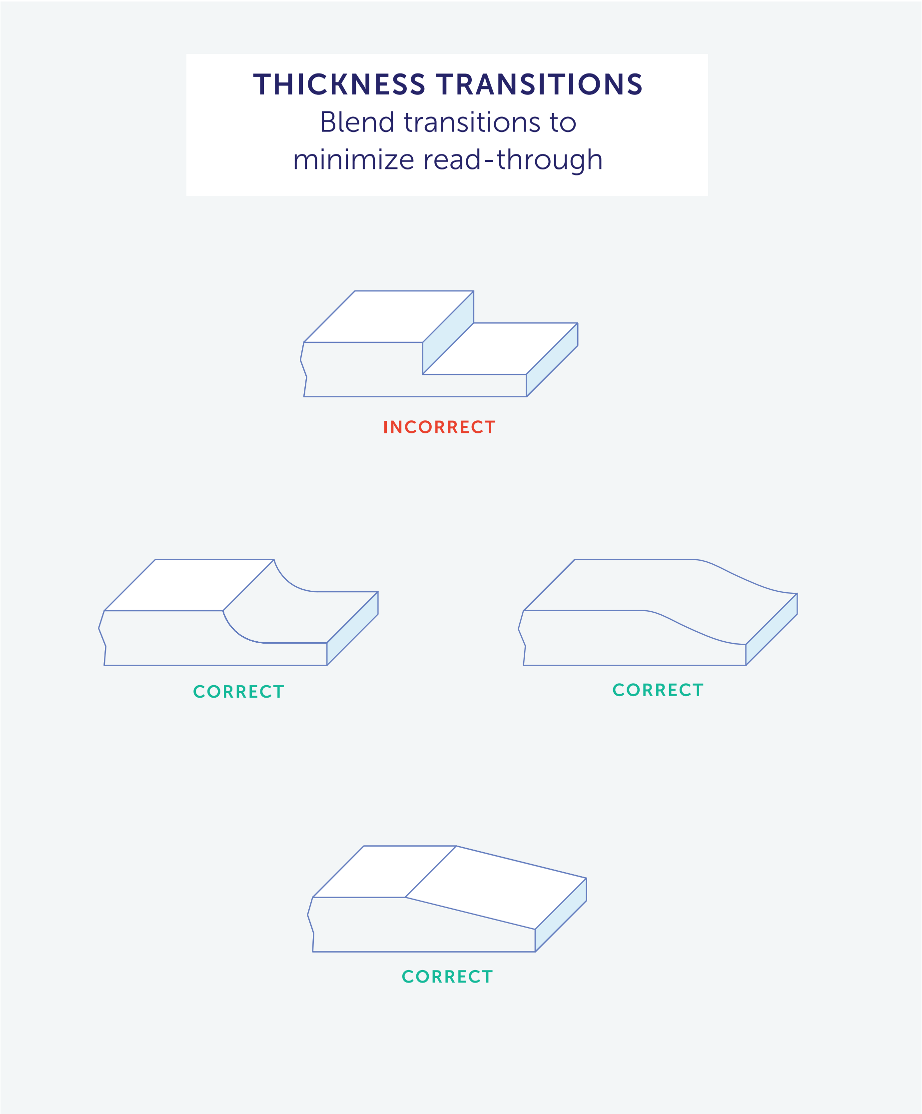
Transitions

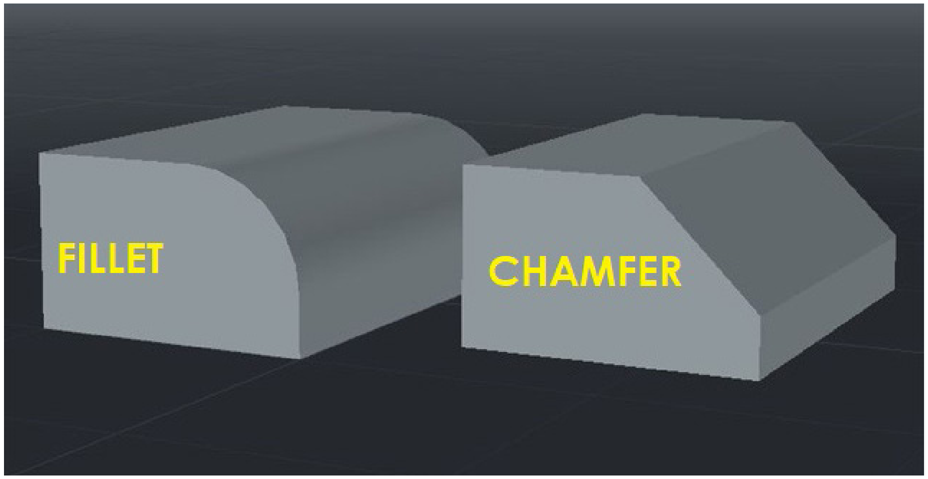
Ⓒ Transitions

Best practice is to smooth, or blend, transitions between areas with different wall thicknesses to minimize stress concentrations that can result in part failure.

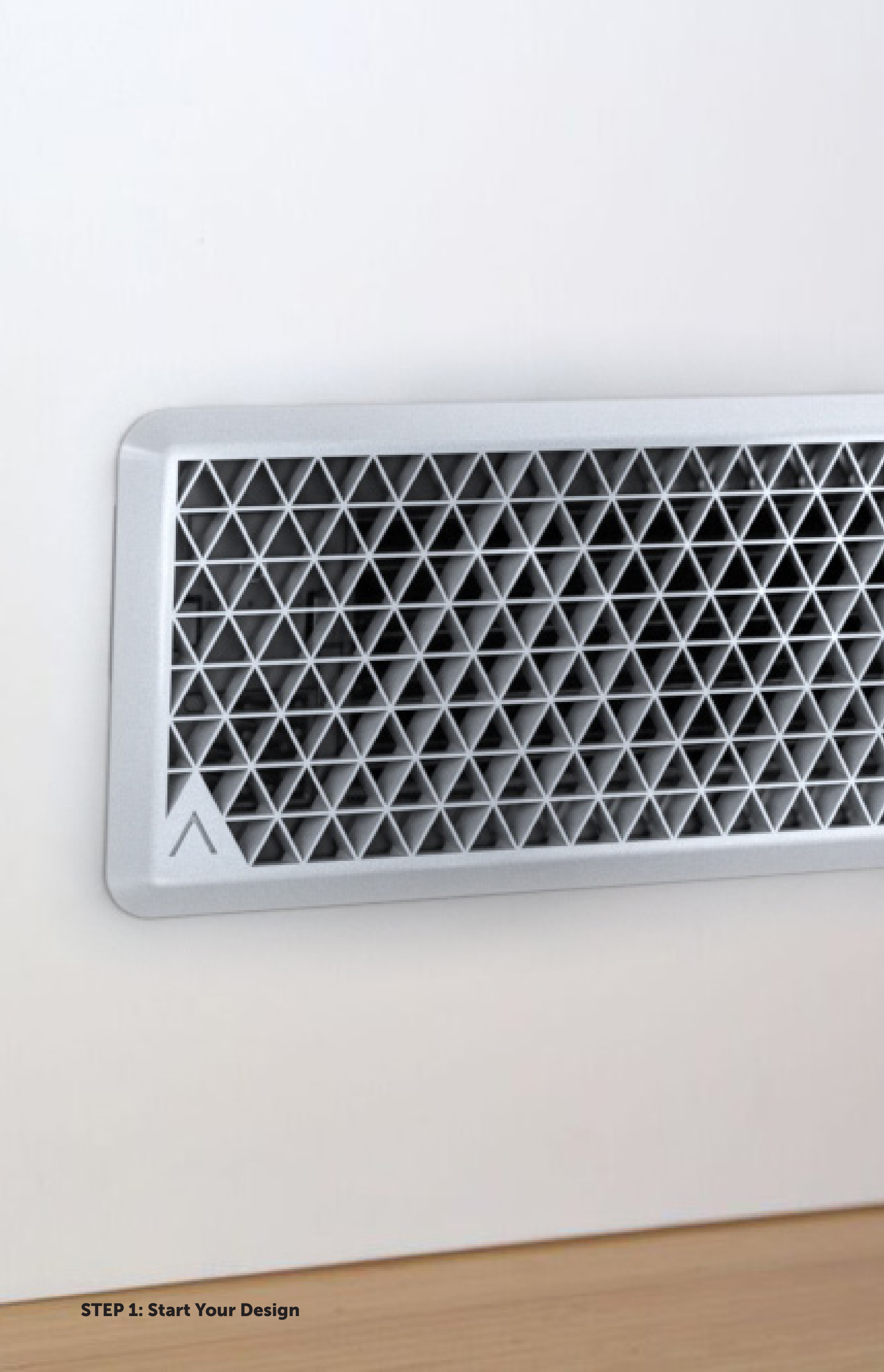
There are two main ways to achieve these transitions: chamfers and fillets.

- **Chamfers** are angled edges where two surfaces meet.
- **Fillets** are rounded corners or edges.





Smooth transitions aren't the only way to avoid stress concentrations, however. Using rounded corners instead of sharp ones can help.



Corners

Ⓓ Corners

Sharp edges don't just concentrate stresses. They also increase part costs because they require molds that are made with electrical discharge machining (EDM).

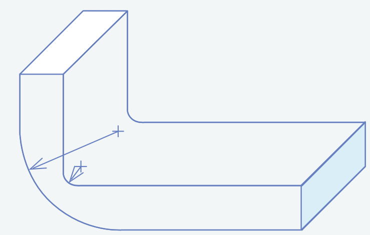
Sharp corners are a good place for parting lines, as we'll see later in this guide, but it's best to use them wisely and, where possible, use rounded corners instead.

Rounded corners limit stress concentrations and minimize differences in shrinkage as the plastic material cools. They also help to control tooling costs and enable the molten plastic to flow more readily throughout the mold.

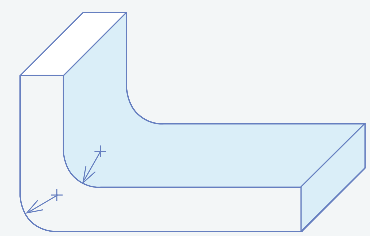
When you apply rounded corners to your part design, follow these guidelines:

- Make the internal radius at least 50% of the wall thickness
- Make the external radius the sum of the inside radius and the wall thickness
- Start the internal and external corner radii from the same point

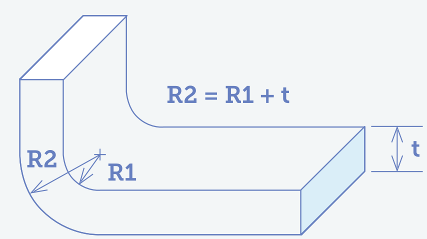
CORNER DESIGN
Internal and external corner radii should originate from the same point



TOO THIN



TOO THICK

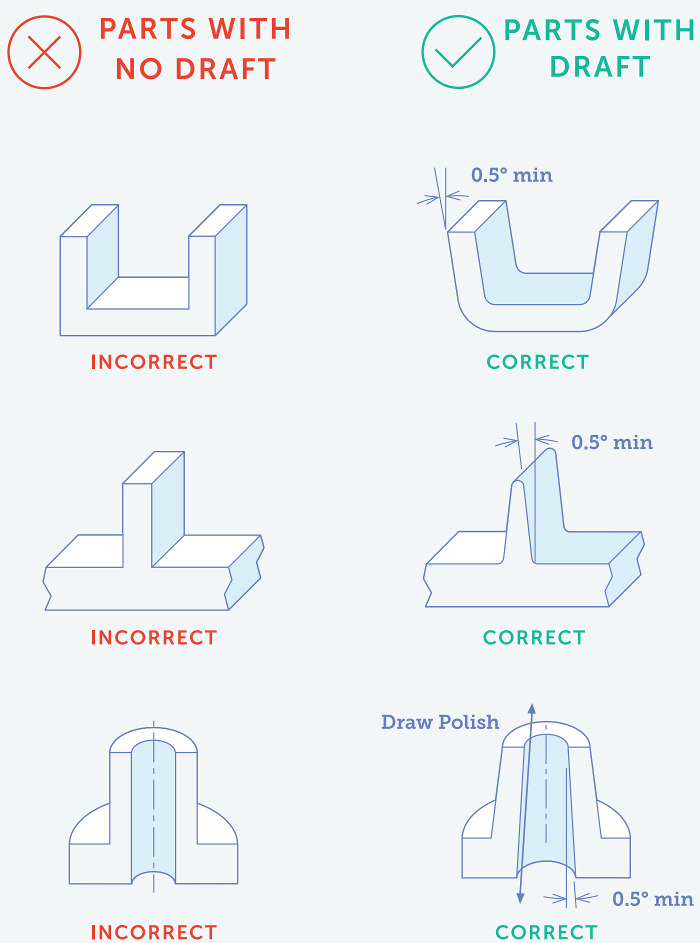


CORRECT

Draft

E Draft

Draft is an angle you apply to vertical walls so that parts will eject cleanly from the mold. This taper can also decrease tool wear and reduce cooling times, both of which help to control costs.



COMMON DRAFT GUIDELINES

When specifying the amount of draft to apply, consider the following.

- The resin you're using
- The standards you're following
- The part's finish
- The mold's construction

RESINS

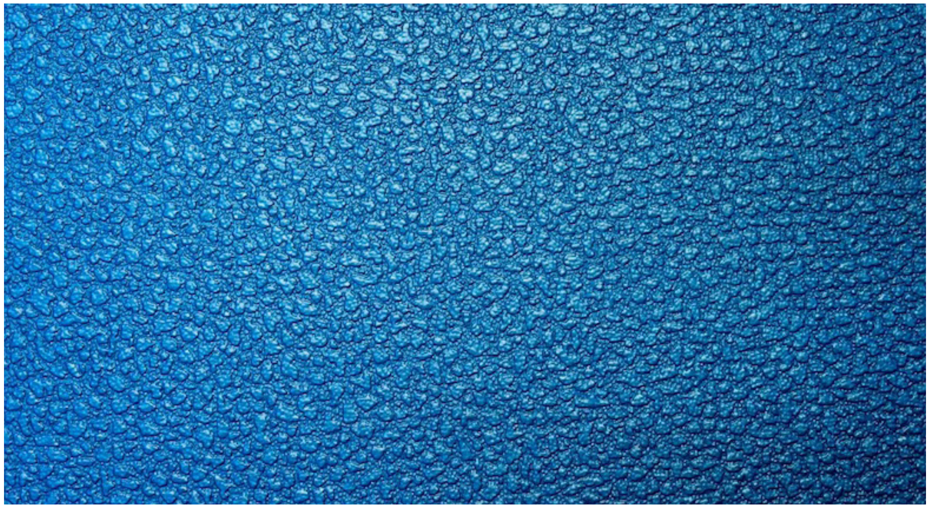
Draft angles vary by the type of resin, or injection molding material. There are also different systems, or standards, that define the amount of draft to apply.

STANDARDS

Trade associations and companies define standards and designations for injection molding finishes, which include textures and polishes.

- The Society of the Plastics Industry (SPI) and the Society of German Engineers / Verein Deutscher Ingenieure (VDI) are two of these trade associations.
- Mold-Tech (MT) and Yick Sang (YS) are companies that have developed their own standards.

This is why you'll hear terms like "SPI finishes" and "Mold Tech textures" during DFM discussions.



FINISH

Draft is related to the texture of your injection molded parts. During injection molding, the texture that you want is transferred from the mold to part surfaces.

Smoother finishes require less draft and finishes with heavier textures require more draft.

- **Smooth Finish:** 1 to 2 degrees (typical)
- **Light Texture**: 3 degrees
- **Heavy Texture:** 5 degrees or more

In general, it's best to add 1.5° of draft per 0.001" or 0.025 mm of textured depth. SPI, VDI, MT, and YS publish tables with recommended draft angles.

Polish isn't a function of draft, but it's worth mentioning here because — as with texture — the finish for your part is imparted by the finish of your mold.

MOLD CONSTRUCTION

Finally, remember to set the draft angle based on the mold's draw, or how it separates. Otherwise, the part may not stay in, or stick to, the half of the mold that contains the ejector system. Consider both the draft on the part's vertical walls and on part features.

For example, in the case of a rectangular part with four through-holes, drafting the holes toward the cavity could cause parts to stick there instead of to the core. A better approach would be to draft the holes toward the core where the ejector system is located.

Ribs and Bosses

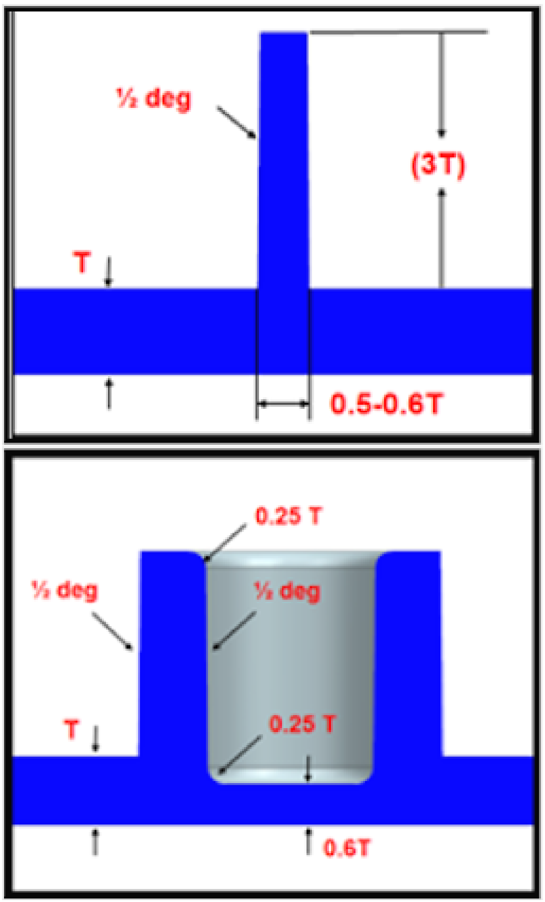
F Ribs and Bosses

Injection molded parts have thin walls to support faster production speeds and longer mold life. Yet thin-walled parts may lack sufficient strength. Ribs and bosses increase part strength, but there are some key rules to follow when adding them to your design.

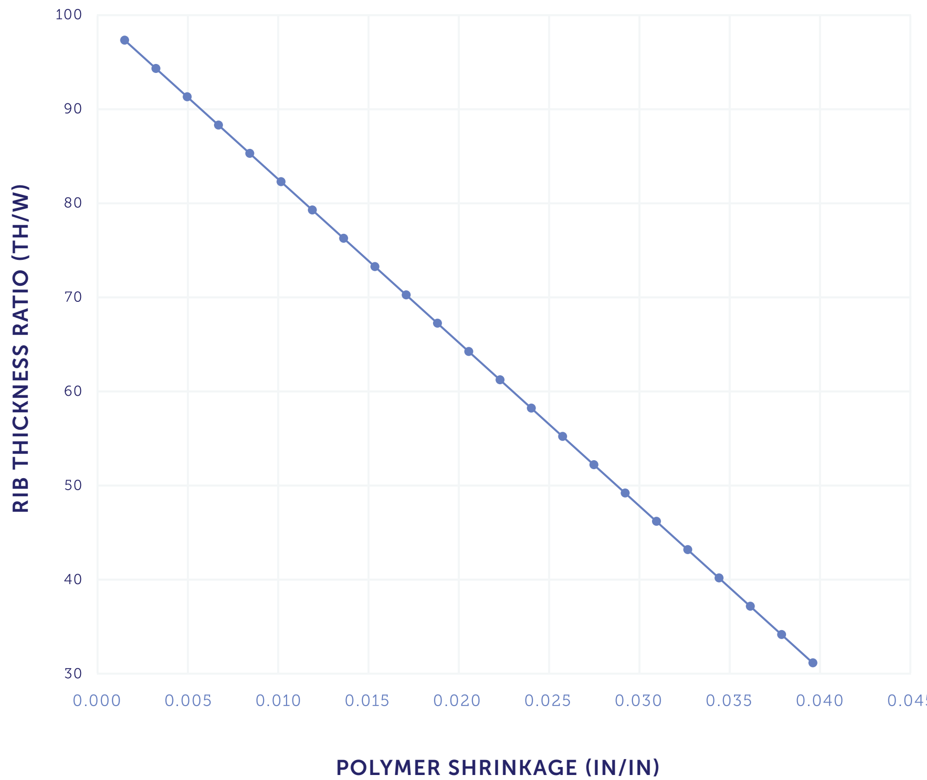
RIBS

Ribs are vertical features that add structural integrity and increase load-bearing capacity.
Thick ribs are prone to shrinkage, however, as this chart shows.

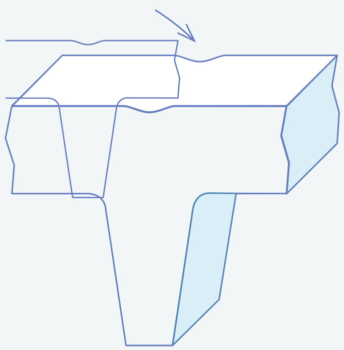
GOOD DESIGN OF RIBS & BOSSES



RIB THICKNESS VS SHRINKAGE



If a rib is too thick, sink marks may appear on its opposite side and mar the part's finish.



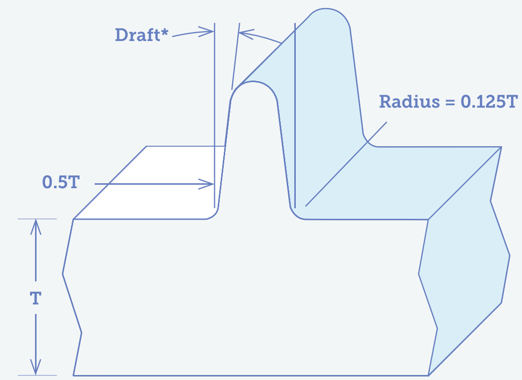
SINK OPPOSITE THICK RIB

To avoid these problems, follow these best practices:

- **Wall Thickness:** Make the rib's wall thickness 50% to 60% (0.5 to 0.6 T) as thick as the nominal wall.
- **Fillets:** Add a fillet to the bottom of the rib. Keep the radius of this fillet near 0.25T-0.5T, where T is the nominal wall thickness. Don't exceed 0.010 inches for the radius.
- **Height:** Keep ribs as short as possible and do not exceed 2.5T. If the height of a rib is a problem, use multiple shorter ribs.
- **Draft:** Apply draft to ribs and use at least 0.5 degrees per side.

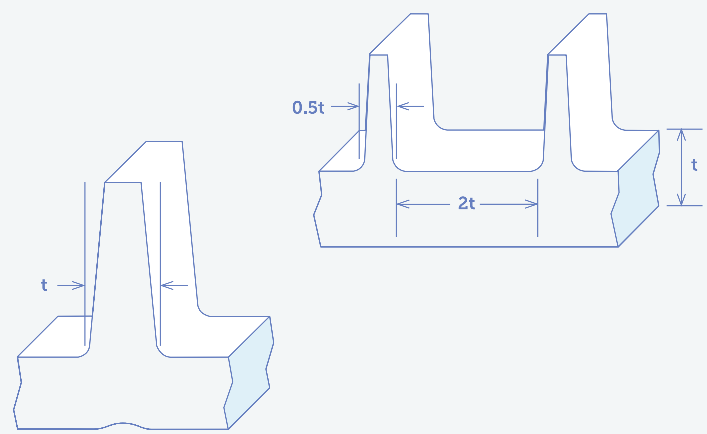
RIB DESIGN GUIDELINES

*Minimum 0.5° per side



MULTIPLE RIBS

Replace large problematic ribs with multiple shorter ribs



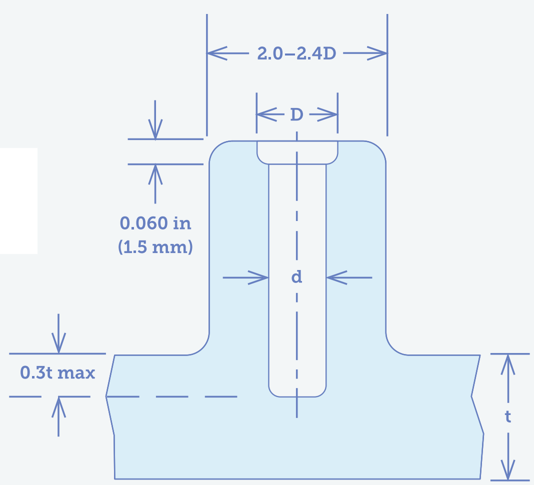
BOSSES

Bosses are vertical features that support assembly and increase a part's structural integrity. They can accommodate fasteners such as screws and, as with dowel pins, smaller bosses can be inserted into larger bosses within walls.

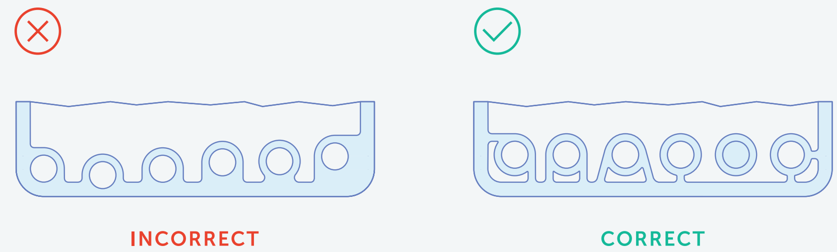
When designing bosses, remember the following:

- **Location:** Place bosses where more structural integrity is needed, such as at screw slots.
- **Diameter:** Don't make the holes too small — they will shrink as they cool.
- **Thickness:** Avoid sink marks by making the boss thickness no more than 60% of the overall wall thickness.
- **Walls:** Consider how bosses will attach to walls — they need to align properly.

TYPICAL BOSS DESIGN

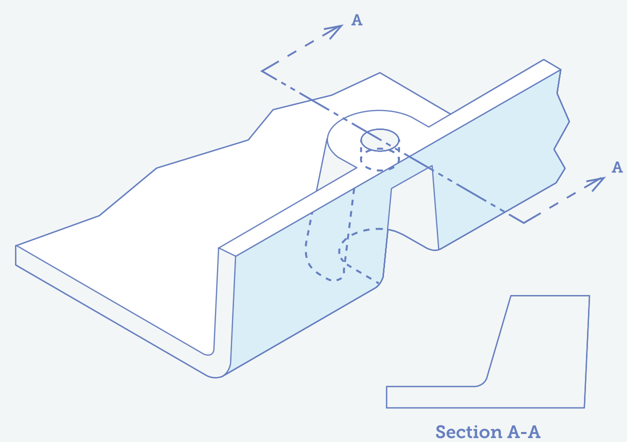


CONNECTING BOSSES TO WALL



BOSS IN ATTACHMENT WALL

Open bosses maintain uniform thickness in the attachment wall



Tolerances

G Tolerances

Injection-molded parts that are used in larger assemblies need to have correct and consistent dimensions. Because some degree of dimensional deviation is expected in any process, designers need to define a part's acceptable dimensional variations or tolerances.

There are two main types of injection molding tolerances.

- **Commercial tolerances** are less precise, require lower-cost molds, and produce lower-cost parts.
- **Fine tolerances** are more precise, require higher-cost molds, and produce higher-cost parts.

Like wall thickness and draft, dimensional tolerances vary by resin. There are also different tolerances for overall sizing and for specific part features.

- **Dimensional:** Overall sizing of the part
- **Straightness or Flatness:** Covers the general warping of large, flat areas
- **Hole Diameter:** Larger holes need larger tolerances because of increased shrinking
- **Concentricity/Ovality:** A large cylindrical part with a thin wall can shrink unevenly

For mating parts, consider the **tolerance stack-up** as well. This refers to how all of the components in an assembly, and their individual tolerances, must fit together.

For example, an assembly has three parts, each with a screw hole. The hole in each part is within tolerance, but the holes must align so the screw can pass through all of them.

Parting Lines

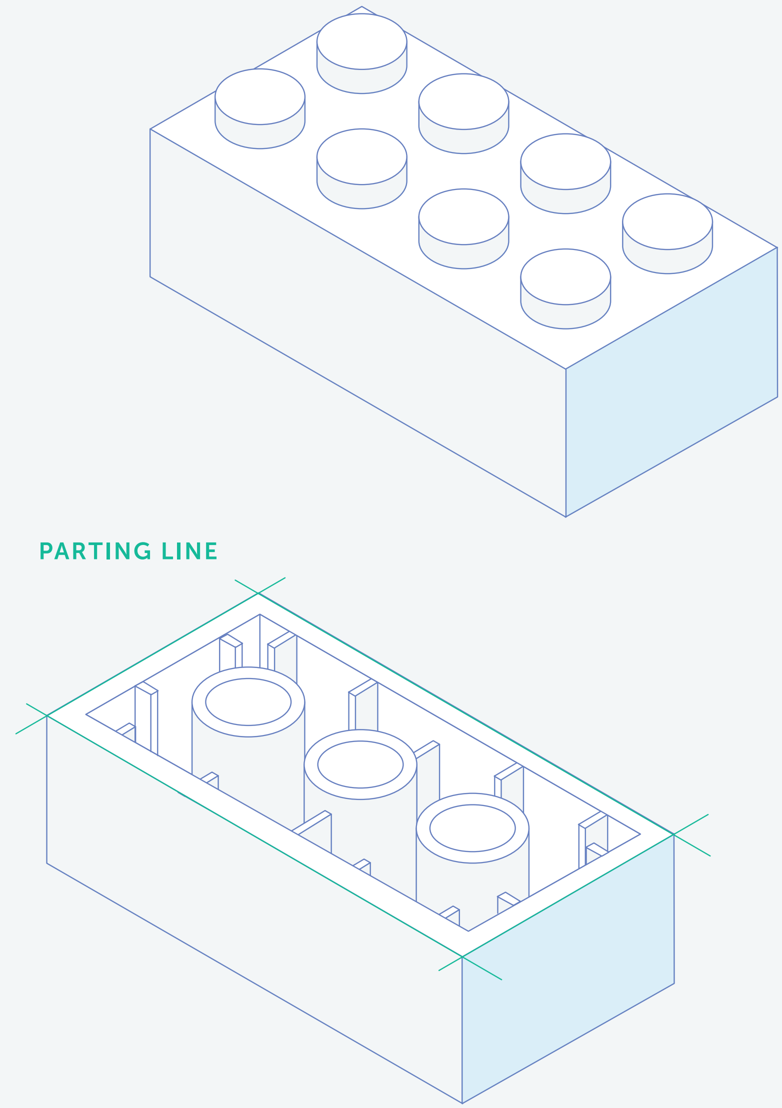
Ⓗ Parting Lines

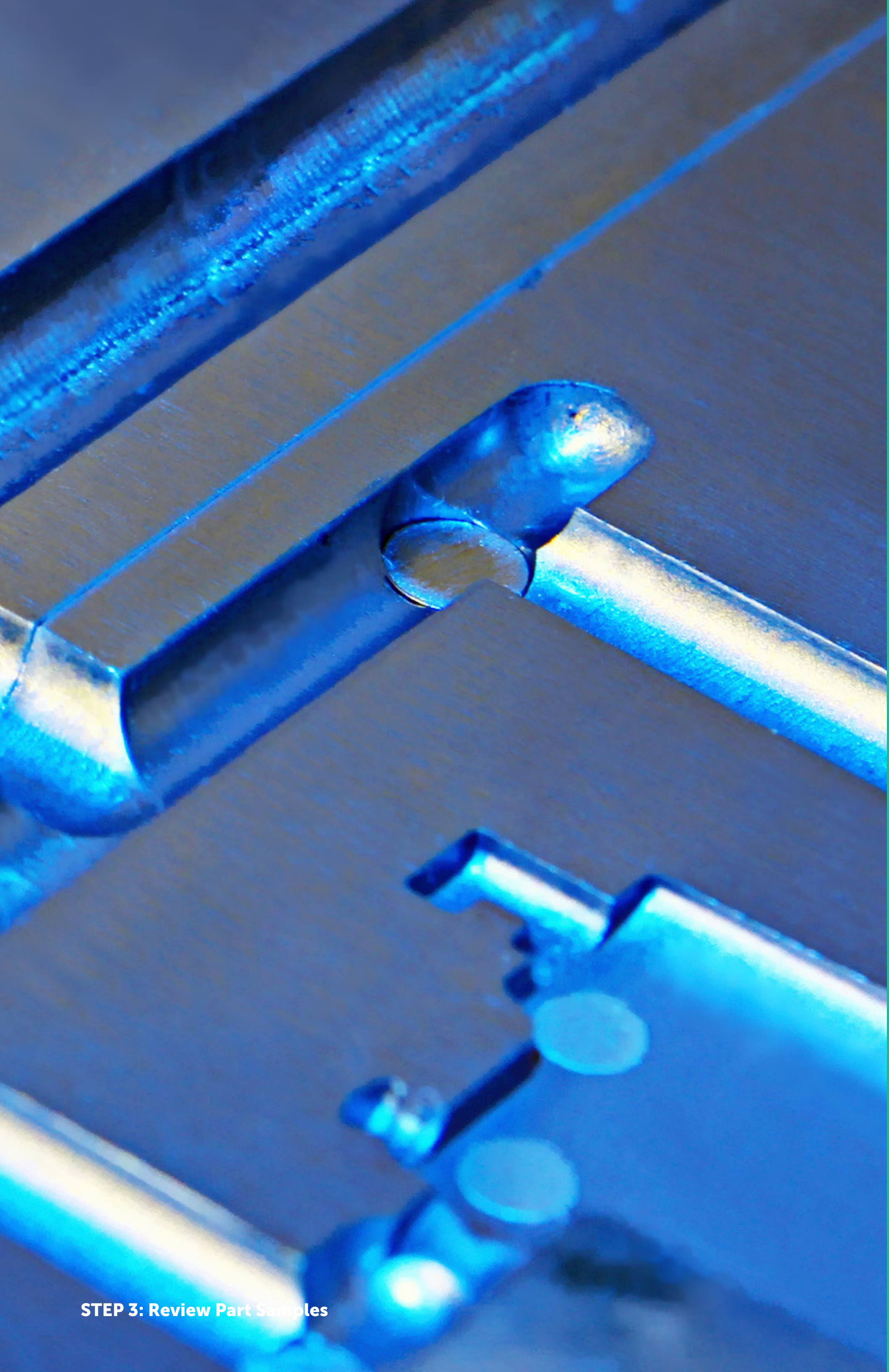
Injection molds have a parting line where the tool opens and closes. Sometimes, the parting line is right down the middle of the injected molded part. That's usually what designers think of when they envision a parting line, but that's not always the case – or even a best practice.

Take, for example, a LEGO® brick. The parting line is not down the top-middle of the brick, where it would be very noticeable. Rather, the parting line is along the brick's bottom edges so that you have to turn over the brick to see it.

Sharp edges concentrate stresses but they're a good place for parting lines because they simplify mold construction, which helps to reduce project costs. Avoid filleted surfaces, however, because they don't work well with parting lines:

- Filleted surfaces require a tight tolerance mold, which increases costs.
- They increase the risk of flash, an injection molding defect that can occur when the two halves of the mold don't come together cleanly.





Gates

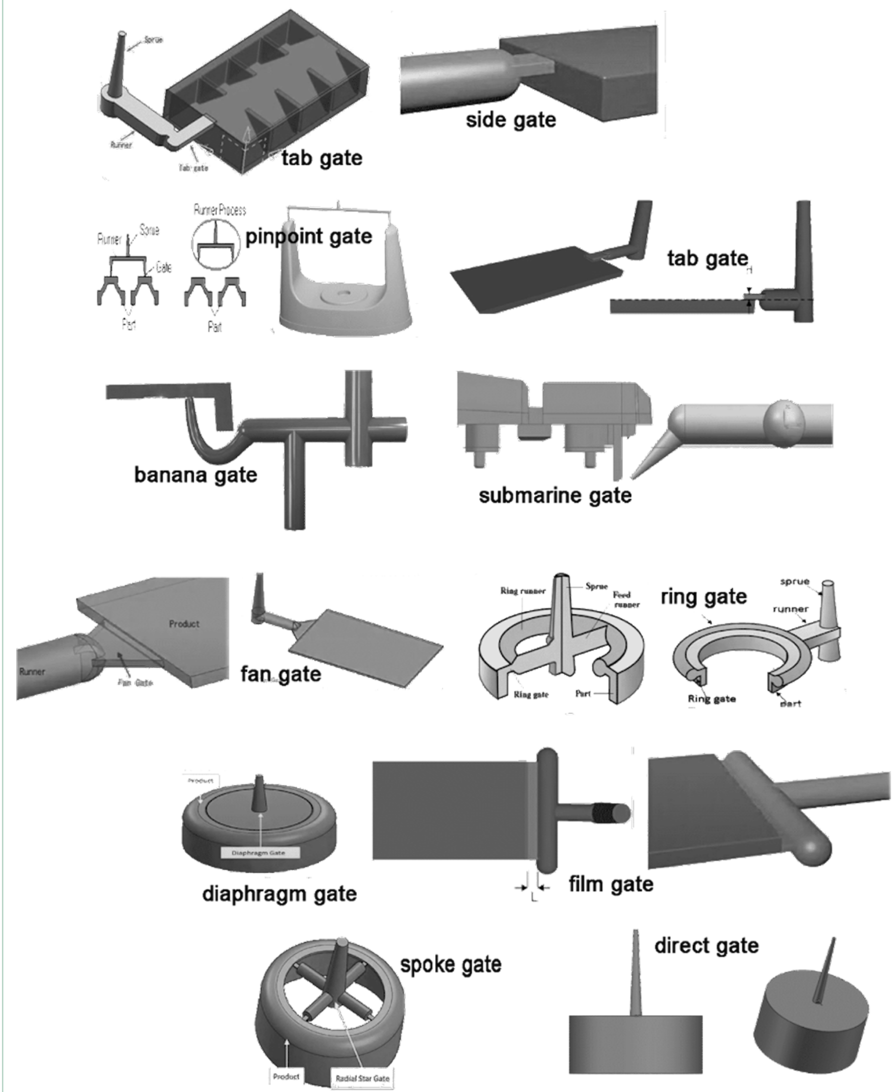
Gates

Gates are openings in an injection mold that allow molten plastic to enter the cavity.

- **Gate size** is important because larger parts need larger gates to support higher-volume flow.
- **Gate location** matters because gates can determine where warping occurs, or where there are weld lines, sink marks, voids, or other injection molding defects.

Designers also need to consider gate location because gates leave plastic protruding from the part's surface. This extra plastic gets trimmed, but it's best to put gates where any small marks that are left behind are least noticeable. Often, gates are placed along the mold's parting line.

Trimming provides a way to categorize gates, but specific types of gates are recommended for parts with specific part features. Some gates require manual trimming. Others support automatic trimming where excess material is sheared during part ejection.



MANUALLY TRIMMED GATES

- **Edge or standard gates** are recommended for flat parts. The gate's cross-section is rectangular and can be tapered.
- **Fan gates** have a large opening with a variable thickness. They permit the rapid filling of large parts and fragile mold sections.
- **Tab gates** are used with thin, flat parts that require low shear stresses. A tab-like feature confines these stresses to the gate area.
- **Direct or sprue gates** are used with large, cylindrical parts. A sprue feeds material directly and rapidly into the cavity.
- **Disc or diaphragm gates** are used with round or cylindrical parts that require concentricity. Typically, these injection molding gates are difficult to remove and expensive to trim.
- **Ring gates** allow material to flow freely before moving into a uniform, tube-like extension for mold filling.
- **Spoke gates** are round gates that have a cross in the middle. They're used to produce tube-shaped parts, but perfect concentricity is difficult to achieve.

AUTOMATICALLY TRIMMED GATES

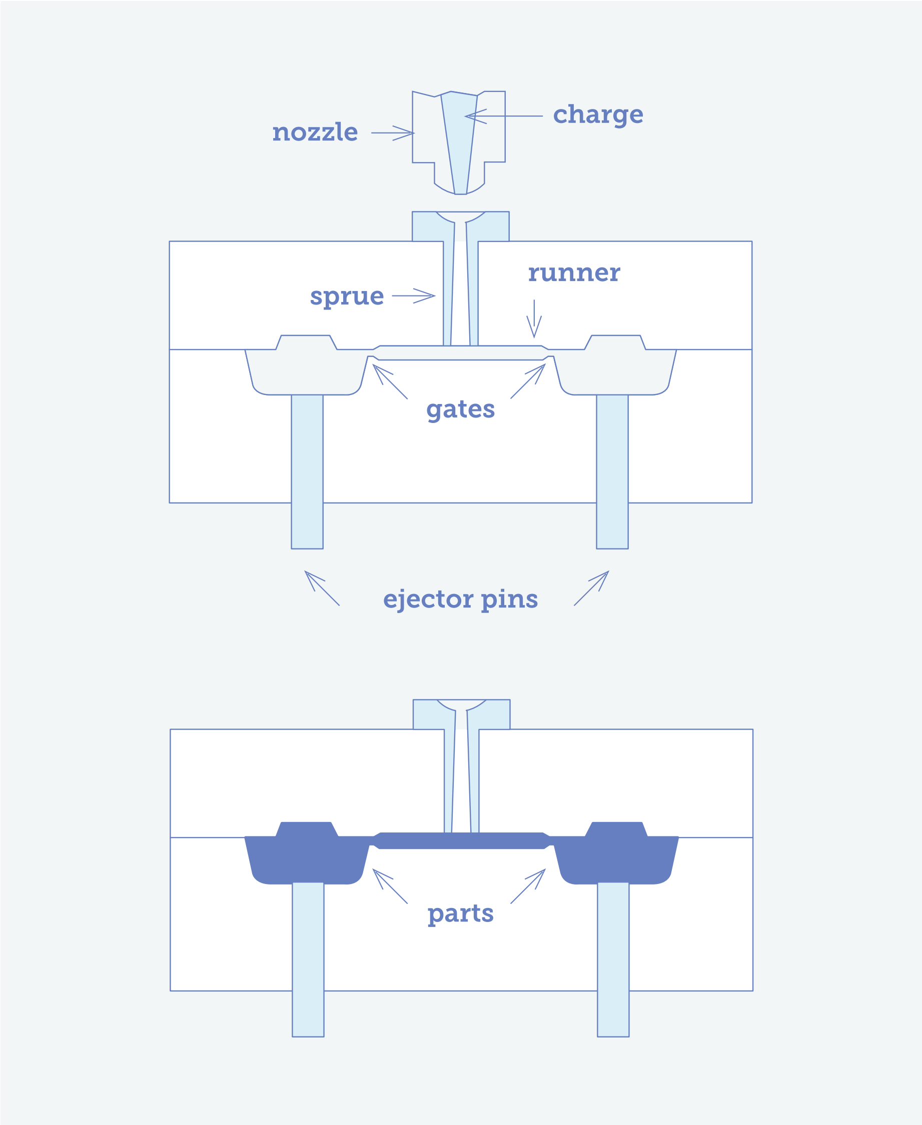
- **Hot tip gates** support conical or round shapes that require uniform flow into the mold cavity. They are used in hot runner systems that keep the plastic in a molten state until it enters the cavity.
- **Submarine or sub gates** have a tapered channel and can help hide gate blemishes. These openings are sometimes called tunnel gates.

- **Pin gates** are used with fast-flowing resins and when a part's cosmetic appearance is important. Pin gates are used typically with parts that cannot have vestiges on either side of the parting line.

Ejector Pins

Ⓙ Ejector Pins

Ejector pins are used to eject a part from a mold after cooling is complete. Sometimes, however, the mold sticks, which causes marks from the ejector pins to appear on the finished part.



For best results, locate ejector pins on part surfaces that are not visible. There are other guidelines to consider as well — some are a function of the mold-making process, but part designers need to understand them in order to evaluate tooling.

- Distribute the ejection force as evenly as possible to prevent part deformation.
- Apply the ejection force to the area of the part with the greatest strength and rigidity.
- Avoid placing ejector pins on thin areas or sloped surfaces.
- Locate ejector pins away from the sliding track that the mold uses.
- Use an ejector mechanism with sufficient strength and wear resistance.

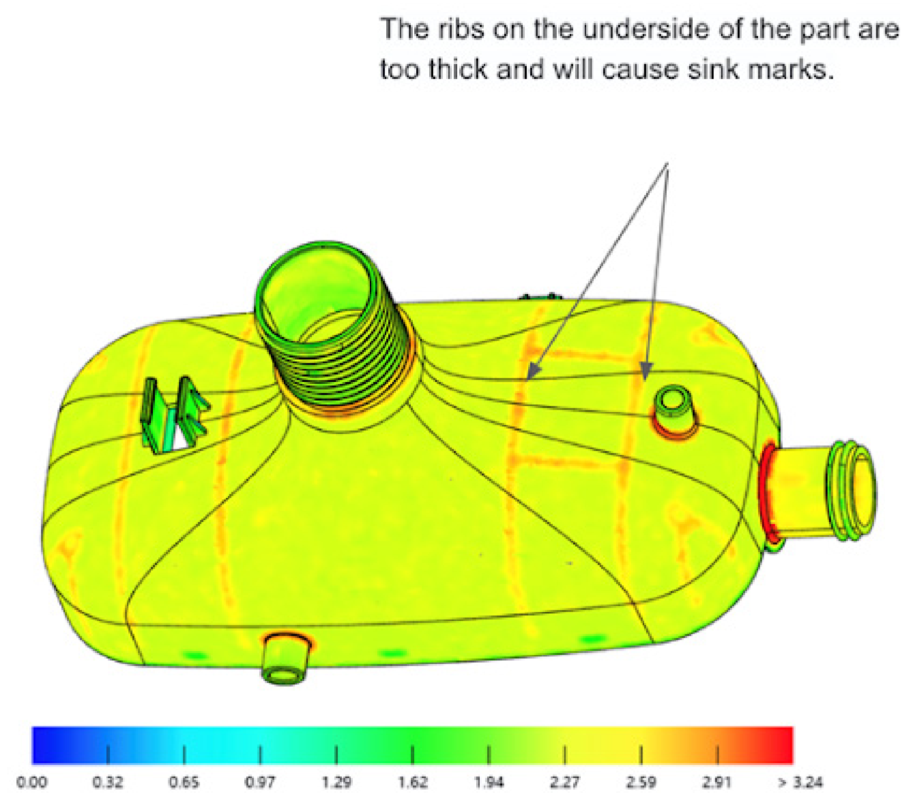
Now that you've started your part design and defined some key characteristics, it's time to upload your design and get expert DFM feedback from Fictiv.

Step 2: Upload Your Design and Get Expert DFM Feedback

fictiv.com/login

Step 2: Upload Your Design and Get Expert DFM Feedback

Fictiv provides DFM feedback along with your request for a quote. Just create a free Fictiv account, upload your CAD file, and then review and incorporate the feedback using the 3D visualization tool built-in to the Fictiv platform.



Most manufacturers won't provide DFM feedback until you place an order, but Fictiv provides a full DFM analysis for free along with your quote. At this stage, the aim is to discover all of the potential issues or risks for mold making and injection molding.

Here are a few common DFM questions that Fictiv's engineers will answer when you work with us:

- Will the location of the gate cause cosmetic defects?
- Will the parting line leave a noticeable mark?
- Will ejector pins noticeably mar the part or risk damage?
- How will plastic flow into the mold and form what's known as a knit line (weld line), where two flow fronts meet?
- Are there additional risks, such as hard-to-achieve tolerances?

The DFM stage is critical because when your part is still being designed, small modifications typically take just a few minutes or hours to complete. After your tooling is cut, however, it can take at least a few days and hundreds of dollars just to make a small tooling change.

By working with Fictiv and incorporating the expert DFM feedback that you receive from us, you'll be ready for mold-making and part sampling. You'll also get access to a highly-vetted network of local and overseas injection molders, as well as a dedicated U.S. service team.

Step 3: Review Part Samples

Step 3: Review Part Samples

During Step 1 of the injection molding design process, you'll consider how the injection mold that will be used to produce your parts can affect part costs and quality. During Step 2, Fictiv reviews your design for manufacturability.

In the final step of the injection molding design process, a tool maker will create your mold and an injection molder will send you part samples. Because your part design may change based on the trial samples that you receive, there are a few more steps to consider.

- **A** Mold Making
- **B** Sample Parts
- **C** Begin Production

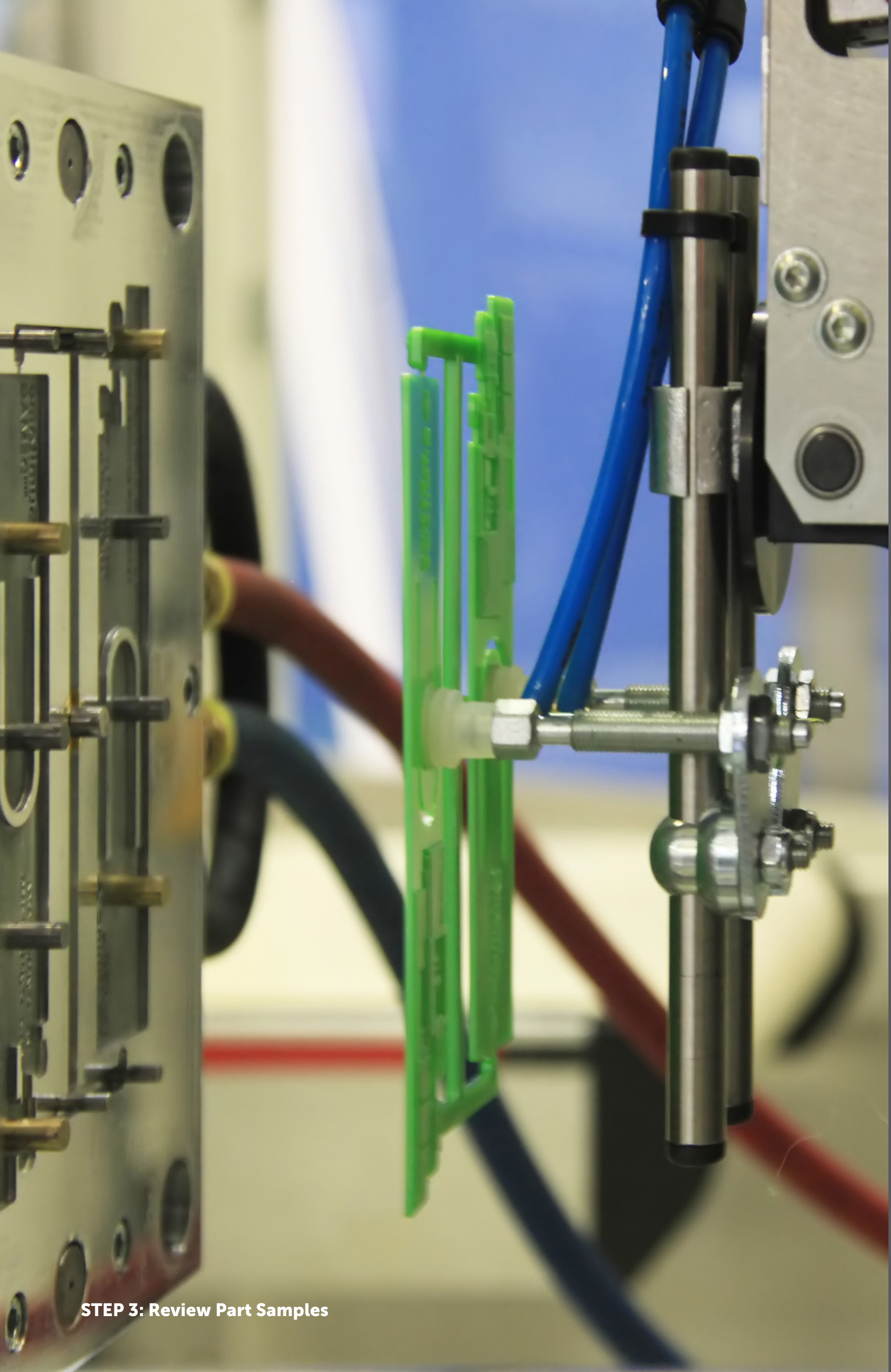
Mold Making

Ⓐ Mold Making

After you submit your part design for tooling, a mold maker will cut the steel or aluminum that's used for the core and cavity. Bench workers will then assemble the mold, incorporate off-the-shelf parts such as ejector pins, and test the mold for leaks.

For new product introduction (NPI), Fictiv recommends starting with a single-cavity tool because these molds are less expensive to make and take less time to produce. That's important since your design may change and require new tooling.

When your design is truly final, you can ramp up production by ordering multi-cavity molds or family molds that combine different part numbers into the same mold base.



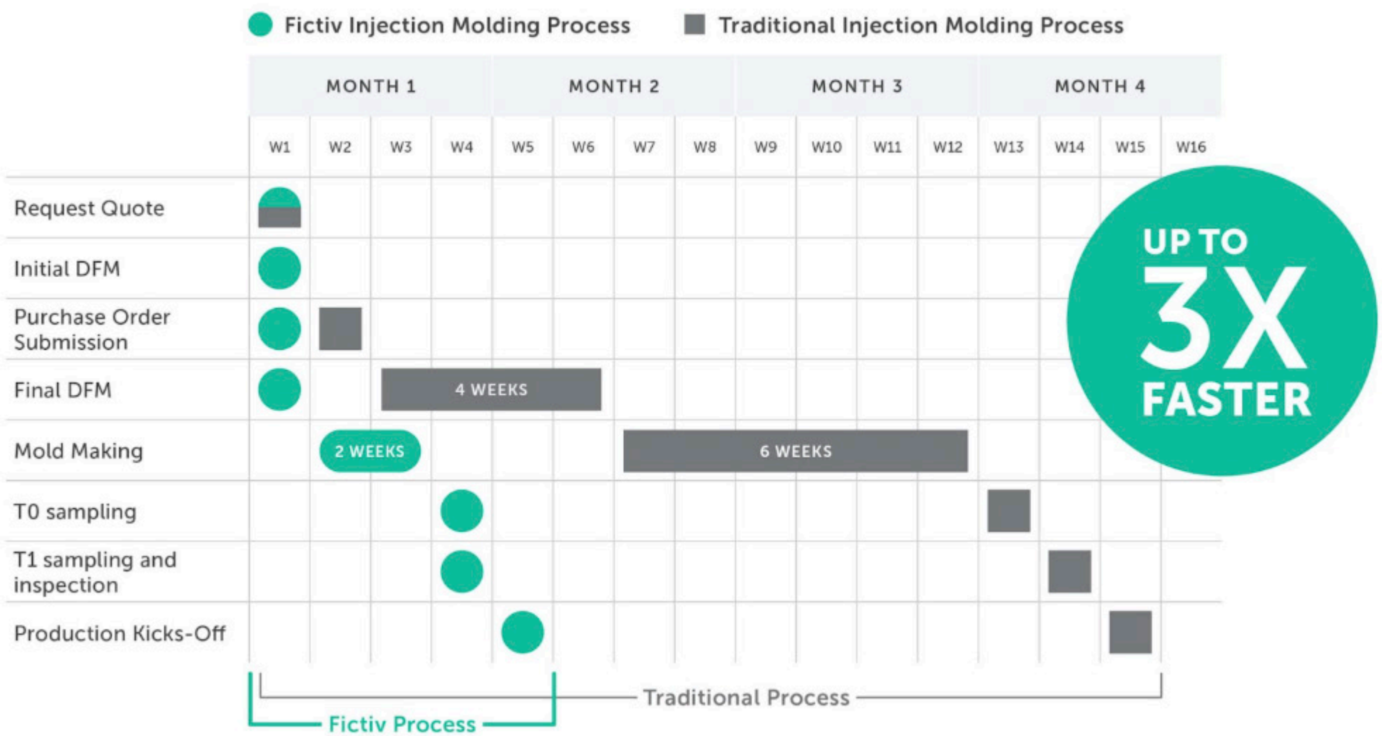
Sample Parts

B Sample Parts

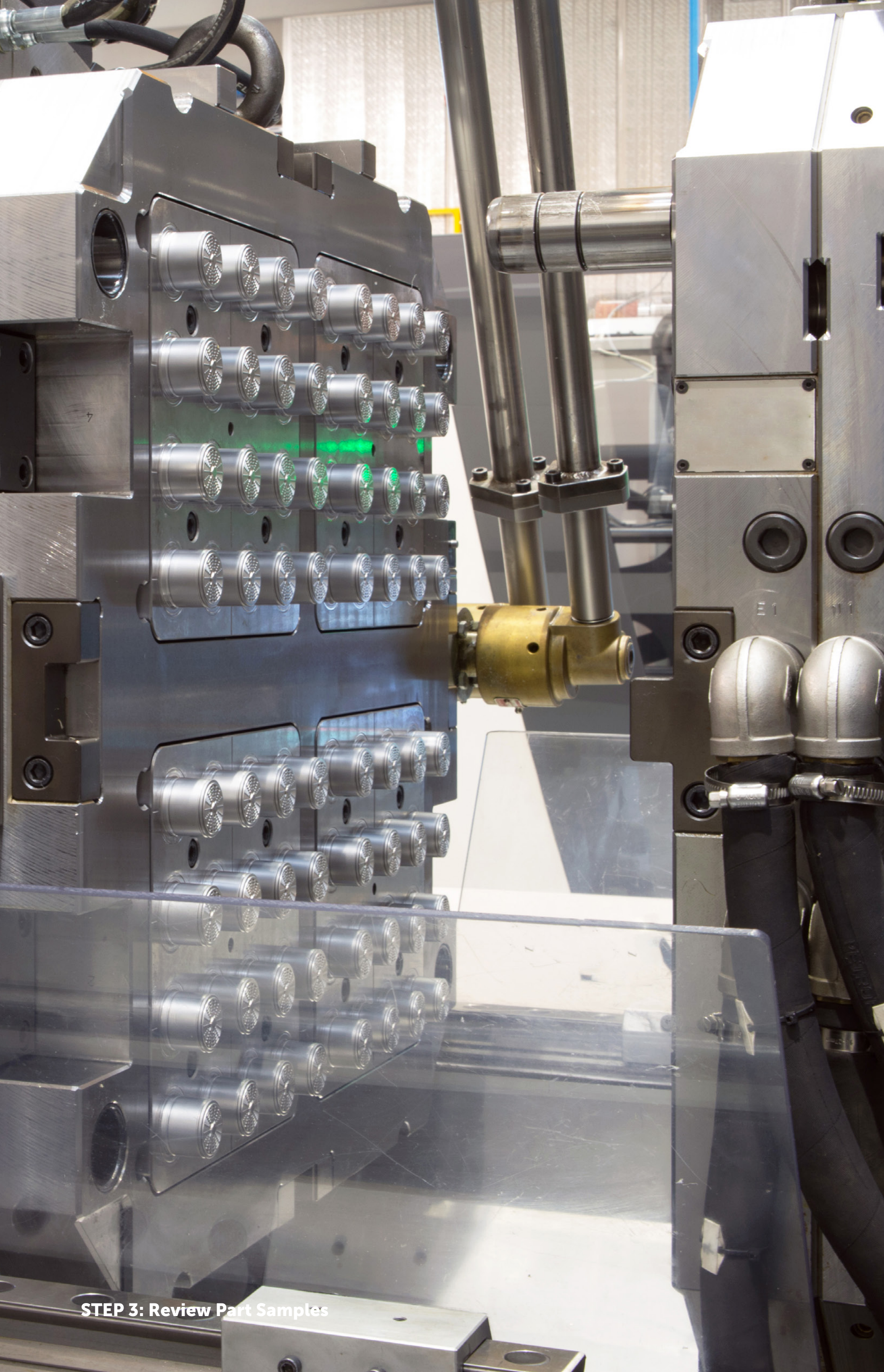
Use the sample parts that you receive to perform functional testing and dimensional measurements. Expect your samples to be within the tolerances you've specified and very close to meeting all of your requirements.

If your part design included a textured finish, your sample parts will include this as well. There are hundreds of texture options, so make sure the light, medium, or heavy texture you've received is satisfactory.

If your sample parts require adjustments, submit an engineering change order. For example, you might need to tighten up some tolerances. Then, when you're satisfied with your samples, it's time for production.



The information in this chart is provided in good faith, based on Fictiv employee experience with traditional manufacturers, and is provided "As Is" without warranty of any kind (express or implied).



Begin Production

Ⓒ Begin Production

After you've approved your part samples, injection molding production can begin. Depending on the tool's construction, you can get tens, hundreds, or many thousands of parts.

Because Fictiv doesn't have minimum order quantities (MOQs), remember that you can ramp up production gradually to match customer demand.

Minimizing Risk During Product Launch

As you've learned from this guide, there's much to consider when designing injection molded parts. But don't worry! Partnering with Fictiv means that we'll shoulder the heaviest parts of the burden. Count on our guided expertise to improve your design and ensure a successful product launch.

Create your free Fictiv account and upload your part drawing at **fictiv.com/signup**

fictiv

Injection Molding

High quality tooling,
done your way.

- Online quotes in 24 hours with free DFM
- Complex geometries accepted
- Support from a U.S.-based team of tooling experts

LEAD TIME | T1 samples as fast as 10 days

QUALITY ASSURANCE

- Standard: SPI Commercial Tolerances & Inspection Reports
- Available: SPI “Fine” Tolerances, Material Certifications, CoCs

TOOLING TYPES & FINISHES | Types: Single Cavity, Multi-Cavity, Family Tooling. Finishes: Glossy, Semi-Glossy, Matte, Textured

TOOLING MATERIALS | P20 Steel, NAK80 Steel, H13 Steel, Stainless Steel, Aluminum

SECONDARY OPERATIONS | Pad Printing, Silkscreen Printing, Painting, Laser Etching, Ultrasonic Welding, Insert Heat Staking, Insert Molding

GEOGRAPHIES | U.S. & overseas options available

INJECTION MOLDING MATERIALS | All commercially available materials + custom materials

PRESS SIZES | 50-3,300 ton machine presses

fictiv