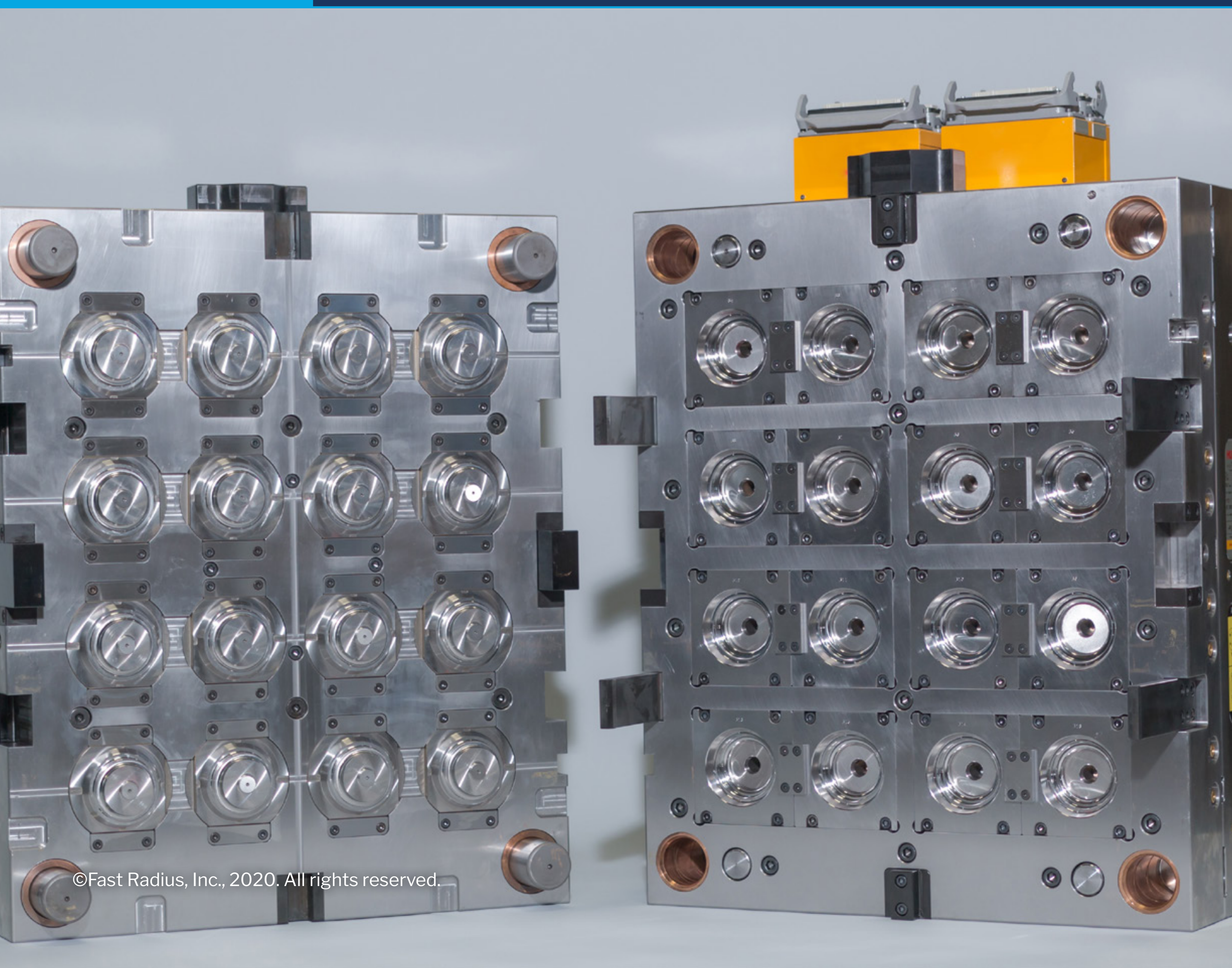
The Fast Radius Guide: Avoiding common injection molding design mistakes

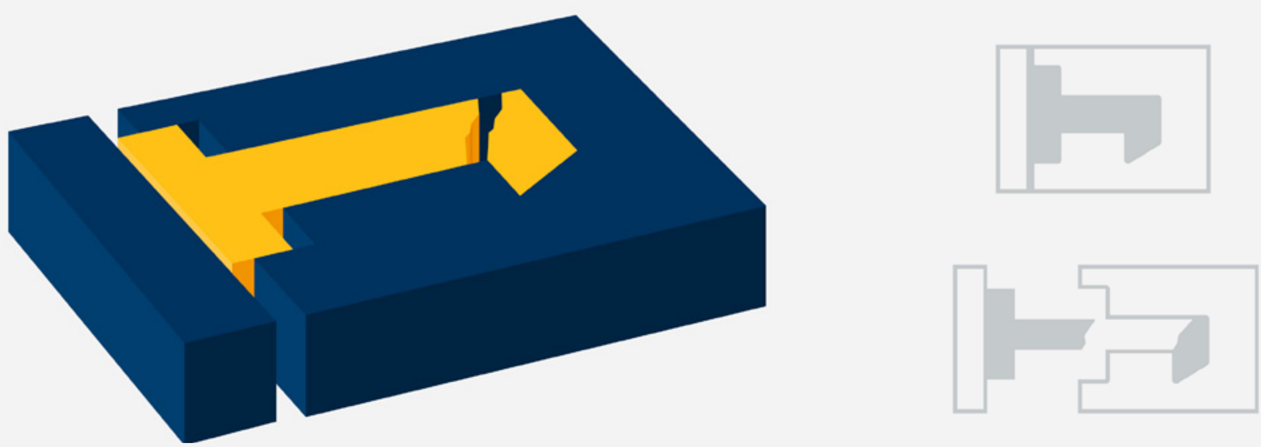


Injection molding design challenges

Injection molding is a manufacturing method that enables the rapid production of identical parts in large volumes. The process involves using a highly pressurized nozzle to fill a variety of molds with molten material.

Typical straight-pull injection molds have two halves — the core and cavity — that separate, allowing the part to be removed efficiently. In order for a straight-pull mold to be designed, a part must be able to be ejected along the line of draw without any obstructions. While this may sound simple, this requirement actually places some significant limitations on engineers and designers.

Design features that obstruct the ability of a part to release from its mold along the line of draw are called undercuts — but they aren't necessarily something that must be avoided. In fact, useful design features like snaps and latches, side ports and holes, vertical threads, and barb fittings are all examples of undercuts, many of which are commonly used in medical equipment and plastic consumer products.



Parts with undercuts cannot be safely ejected without breaking or suffering other damage, but actions can alleviate these issues.

Fortunately, it's possible to create injection-molded parts with undercuts and other complex geometries with an in-depth understanding of injection molding principles like draft and key workarounds like side actions.

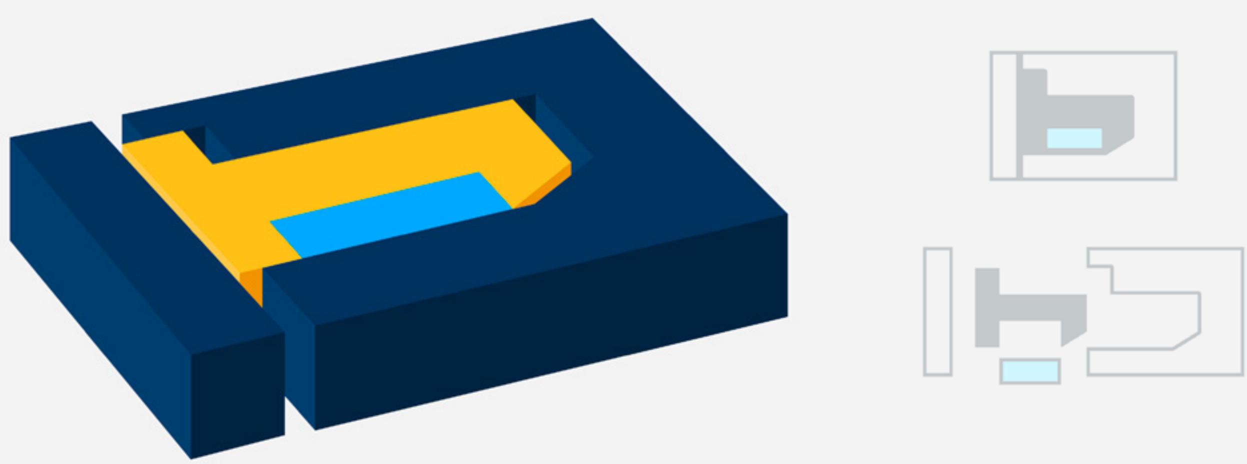
Solutions for undercuts

While there are numerous design solutions for undercuts, each has different applications, benefits, and drawbacks. Some of the most common solutions include slides or lifters. Engineers can also optimize the manufacturability of a part with undercuts by orienting it in the mold so that there are as few undercuts impeding the ejection of the part from the mold as possible.

Actions are special features designed into a mold that allow molding of undercut features, but retract to also allow straight ejection of the part. While actions increase the costs and lead times associated with tooling, the additional expense often pays for itself, by allowing manufacturers to produce viable parts with undercut geometries.

Slides, one of many injection molding actions, are small pieces that can be removed from the injection tool for easier part ejection. When the slide is removed, the two halves of the mold separate, resulting in the ejection of the part from the mold. Unfortunately, because slides have to be carefully custom-made to accommodate any potential undercuts, they can be a very expensive and complex solution.

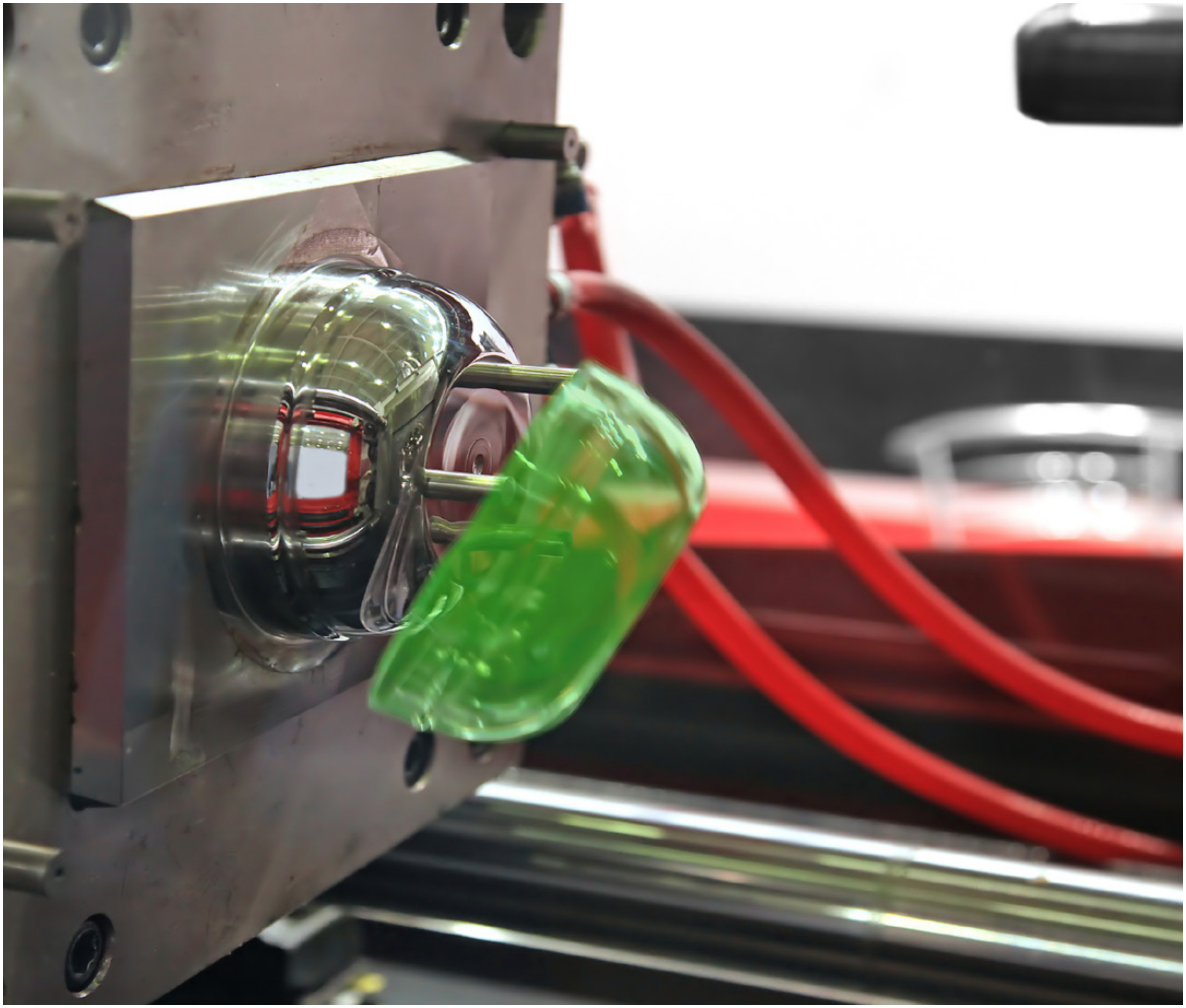
A more simple and less costly solution is to design features that are parallel to the draw line, or the axis on which the injection mold separates. This enables the injection mold to open and the part to safely eject without damage to its features, thereby avoiding an undercut. Unfortunately, depending on the feature, this may or may not be possible.



Slides, as shown above, allow injection molding parts to be printed, even with undercuts.

It’s also possible to create a hole or slot in the mold through which the part will eject without hooking or otherwise sticking in the mold. This allows for the metal in the mold to pass through the hole of the part and properly form the underside of the undercut. Of course, if a part needs to be watertight or has certain aesthetic restrictions, slides will be necessary. However, **it’s usually possible to successfully create an injection-molded part with an undercut — it’s just a matter of selecting the correct action for the given part.**

Actions are special features designed into a mold that allow molding of undercut features, but retract to also allow straight ejection of the part. While actions increase the costs and lead times associated with tooling, the additional expense often pays for itself, by allowing manufacturers to produce viable parts with undercut geometries.



A part being ejected cleanly from an injection molding machine.

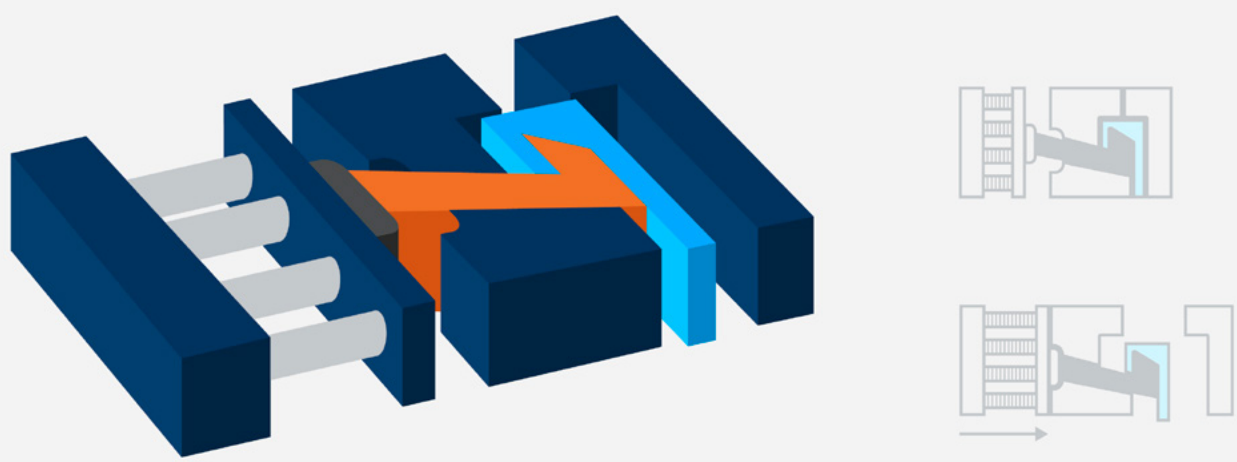
Most common injection molding actions

1. Cams

Cams are simple devices that pull the undercut mold surfaces away from the part, allowing the mold to open. The most common type of cam device uses an angled pin to control the movement of the action, which opens and closes at the same speed as the rest of the mold. Cams allow for far greater geometric complexity in part design without requiring any additional external setup or adjustments from operators during production.

Furthermore, the cam pin itself needs to be monitored over the production lifecycle, as steel is a compressible and flexible metal, which can lead to inconsistent part quality if additional load compresses the pin.

2. Lifters



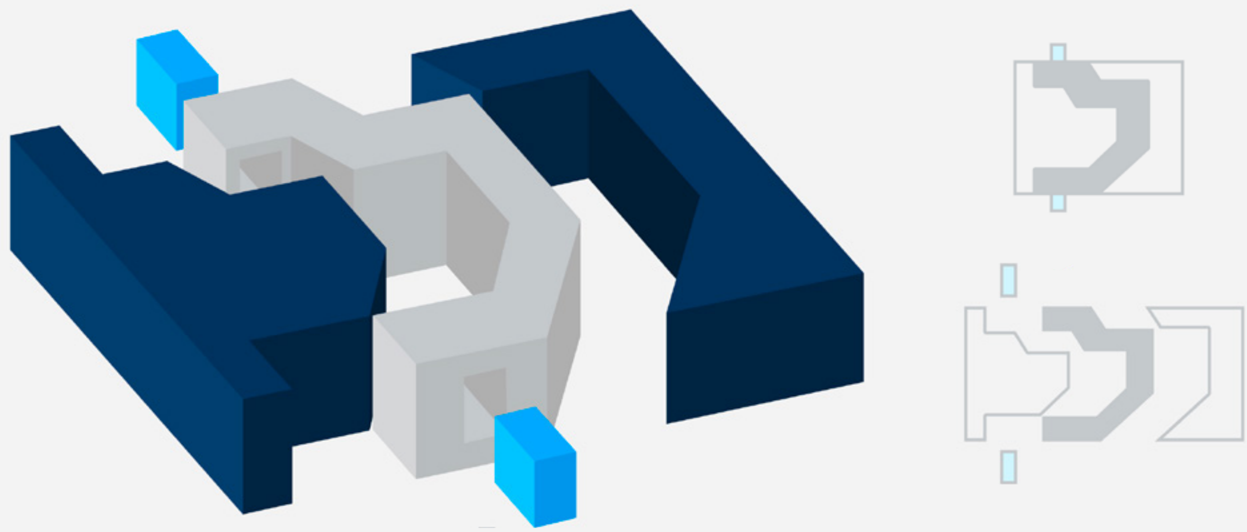
Lifters, as seen in the image above, can help release internal undercuts or internal faces that do not have draft.

Lifters help to release internal undercuts or internal faces with no draft. After the part has cooled, the lifter pushes up at an angle, which removes the undercut section from the mold, and retracts from the undercut during ejection. While lifters can use an angled pin to control the action's movement, they are often driven by the action of the injection molding press pushing against the ejector plates.

A basic action lifter will withdraw simultaneously with ejection — only perpendicular to the line of draw. While the lifter moves at an angle relative to the core mold, it moves sideways relative to the part, which allows the undercut surfaces to be ejected at the same rate that the lifter withdraws. Internal features like draft, ribs, and bosses need to be modified to accommodate the lifter's range of motion.

3. Sliders

In contrast to lifters, action sliders release external undercuts. Similar to cam devices, sliders use angled pins attached to the core mold to pull the slide away from the finished part, allowing it to be removed easily. They can also be solenoid- or hydraulically-driven.



Sliders use angled pins attached to the core mold to pull the slide away from the finished part, allowing it to be removed easily.

4. Unscrewing

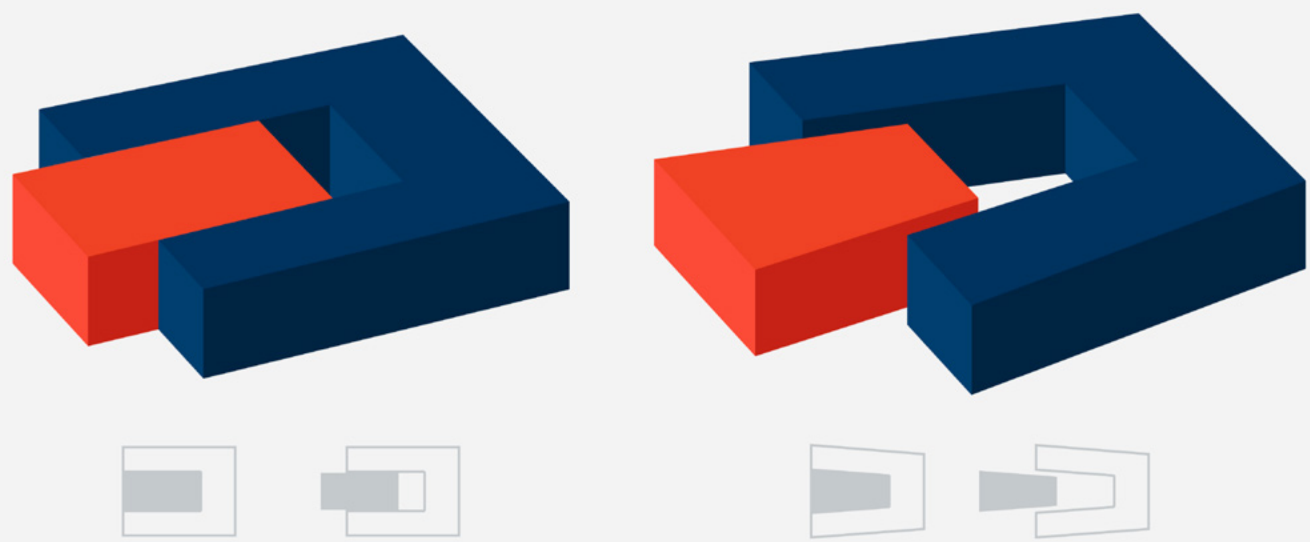
Parts with threaded features — such as screws and other fasteners — are complicated to remove from injection molds. External threads can be easily incorporated into mold design, so long as the threads are perpendicular to the line of draw. Internal threads and non-perpendicular external threads require an external unscrewing mechanism. The action is inserted into the mold before injection, then unscrewed from the part after the material has set.

5. Collapsible core

Collapsible core actions are useful for releasing circular undercuts. Parts are molded directly around the action, the core of which collapses once the material is set. This allows the action — and the part — to be removed easily from the mold. Collapsible core actions can also be used to create threaded features.

Guide to draft angles

Draft, like actions, enables complex injection-molded parts to be ejected free of damage. Draft is the application of a slight taper to every surface in the direction of pull on an injection-molded part — a small and even tedious design element that's vital to the success of a project. To visualize draft, envision an ice cube tray: the slight taper allows ice cubes to slide out easily without falling victim to excessive suction or friction. Parts that are lacking the appropriate amount of draft — or a suitable draft substitute — will not properly eject from the mold.



Draft protects the part from damaging friction, reduces wear and tear during the ejection process, helps ensure a uniform finish, and reduces costs by avoiding the need for complex injection setups.

What's more, draft protects the part from damaging friction, reduces wear and tear during the ejection process, helps ensure a uniform finish, and reduces costs by avoiding the need for complex injection setups. Fortunately, no toolmaker would make a part without draft. For that reason, designing for optimized draft angles doesn't just mean adding draft; in most cases, draft is a given. Rather, optimizing draft means carefully incorporating draft so that it adds to, rather than interferes with, the design and look of the final part.

Need for draft

Not everyone is concerned about draft interfering with the look of the part; toolmakers who are creating parts for industrial or similar purposes, for example, tend to opt for a large amount of draft. That's because, while lots of draft may result in a less aesthetically-pleasing product, it also results in a product that's easy to eject and therefore easy to create quickly and inexpensively without causing substantial wear and tear to the tool.

Designers creating consumer products, on the other hand, tend to be more concerned with aesthetics. For these teams, the goal is often to minimize draft while ensuring the effective functionality of the part. To do so, they generally have two options: incorporating draft so that it appears intentional or incredibly subtle, or strategically placing ejector pins and sleeves to help release the part from the tool.

Ejector pins and sleeves can help push the parts out of the mold, but they're expensive; ejector sleeves, in particular, are costly. However, sleeves follow the geometry of the part, which means they won't leave behind a mark. Ejector pins, on the other hand, can leave marks on the surface of the product. As such, ejector sleeves tend to be preferred by designers who are concerned with the final look of their part — but because they have to be custom-made, they can increase the time and cost of a project. In most cases, however, ejector pin marks can be hidden on the side of the product that the customer won't see, like on the inside surface of an electronics enclosure (TV remote, video game controller, etc).

That's why the subtle and intentional incorporation of draft is favored by many designers. Especially skilled designers can ensure that draft is nearly imperceptible, or can incorporate it in such a way that it appears to be an intentional design element.

Necessary draft angle

The minimum draft angle for any given part is largely driven by the depth of draw, the wall thickness, the material's shrink rate, and the surface finish or texture that is to be applied. **As a general rule, a draft angle of 1.5 to 2 degrees is required for most parts, but draft should average about an additional degree for each extra inch of part depth.** Note that if a part is very small, there's some more flexibility to decrease draft below 1.5 degrees. However, for most parts, 1.5 degrees is the minimum draft requirement.

That said, texture also plays an important role in determining draft. Many injection-molded parts have a leather grain or other texture applied to their surface for aesthetic purposes; however, depending on how deep the texture is, the draft angle may need to be increased to ensure the texture won't be scraped off or damaged during the ejection process.

Automobile interiors are a strong example of strategically-applied draft. Most modern automobile interiors are injection-molded but feature a leather grain texture; a careful eye can discern that the texture depth varies throughout the part in order to accommodate changing draft, but it's barely noticeable. On the other hand, many cheaply-made consumer goods have visibly different textures throughout the part or even texture that has been noticeably scraped off.

Car interior parts often require special attention to draft to make sure the leather grain texture isn't damaged during ejection.

The Fast Radius difference

Achieving the ideal balance of cost, quality, and functionality with injection-molded parts requires an in-depth understanding of draft, undercuts, texture, side actions, and the injection molding process — something not all product teams have. That's where Fast Radius comes in.

Fast Radius delivers superior quality parts at competitive rates, guiding each order from concept to delivery with the attention to detail that ensures our customers don't have to compromise on quality, speed, or convenience.

Our team of highly qualified designers, engineers, and advisors can ensure your part incorporates draft and side actions subtly and strategically so that all aesthetic and functional concerns are accounted for. Fast Radius is dedicated to helping your team adjust and modify its design so that compromises in quality aren't necessary.

If you're interested in learning more about how Fast Radius can help your team optimize part design and deliver better products, faster, contact us today.