

- Straight-pull mold: work piece made of at least two pieces of steel/aluminum, held together with mechanical clamps or hydraulic pressure while the plastic is injected and then pulled straight apart
- Side-actions (cams/slides): added to mold to create openings in the sides or the part or to create more complex features
- Parting line: where the two halves of the mold meet
- Runner system: series of channels for the molten plastic that is extruded and pushed into mold
- Hot tip gate: specialized gate that injects the resin into a face on the mold. Doesn't require a runner or sprue.
- Keep part wall thickness as consistent as possible
- Plastic shrinks as it cools and it cools from the outside in. Can cause internal stresses or voids
- Sharp corners cause stress risers in the part, becoming a potential failure point
- Better to make features thin because if an initial test molding shows a feature too thin, it is easier to machine the mold to make that feature thicker. A feature that is too thick may require recutting the entire mold.
- For straight pull mold, side walls of part are more or less parallel to the mold's directions of pull (direction in which the part will be pulled out of the mold)
- To ensure the parts comes out smoothly, need to make sure the walls are NOT exactly parallel. Need to slant as they're cut into the mold so the part will come out easily
- Without including enough wall angle (draft), drag marks may appear on the part, which is only a cosmetic flaw. Results from the part sticking to the mold as the part is ejected
- For plastic without a surface texture, you can base the draft angles on the height of the feature
 - 2 in wall → 2-degree draft angle
 - 3 in wall → 3-degree draft angle
 - etc.
- Draft angle is applied on each face so you need to keep track of wall -thickness.
- Can start with a good wall thickness but the effects of draft can cause the end of the wall/rib to be too thin for the material or the base of the wall/rib to be too thick
- Rounded corners allow the material to flow more efficiently during injection and also reduce the stresses on the material during cooling, which helps reduce the part's tendency to bow, warp, or develop fragile corners.
- Key to reducing these problems is maintaining thickness through the corner
- Inter radius should be at least half of the wall thickness. Exterior radius should be the inside radius plus the wall thickness
- Using a large-exterior radius without adjusting the interior radius will thin the wall around the corner
- Using a smaller outside radius will create a thickening around the corner that can also obstruct the flow of resin in the mold
- Important to include a radius on edges of interior features to aid the flow of the plastic and to improve the strength of the part. Also helps to reduce wear in the mold itself.
- Warp occurs when the features of a part bend, or the whole part bends, as the material cools.
- Distortion can happen if portions of the part have inconsistent material thickness, sharp corners, thick/thin areas, etc.

- Sink is a type of warp that happens in the middle of a face when the material is too thick. The hot material fills the cavity completely, but when it cools, it contracts toward the center of the feature volume, causing the part to sink.