Module 3. Injection Molding

MECH 392, UBC, Instructor: Ahmad Mohammadpanah

Injection molding involves the solidification of a molten thermoplastic under pressure inside a metal mold. The cavities inside the mold are precisely machined to match the geometry of the intended part. Injection molding is used to manufacture many plastic products, such as, bottle caps, disposable utensils, shoe soles, toys, luggage, electric plugs, toothbrush, furniture, and many other plastic products. Enjoy watching this video:

Plastic Processing Overview (watch videos)

Elements of Injection Molding Machine

Module Objective:

1. Understand and compare the mechanical properties and behavior of thermoplastics to thermosets.

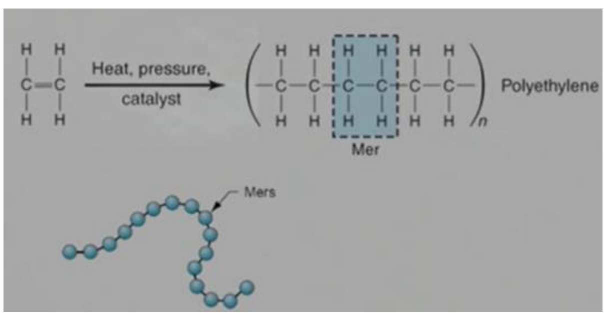
2. Understand fundamental of injection molding process parameters.

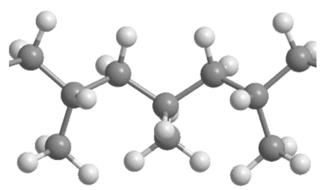
3. Estimate the injection pressure, clamping force, and the cooling time.

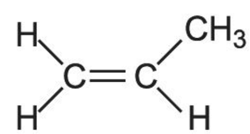
Polymer:

You can order online (https://www.alibaba.com/showroom/polyethylene-pellet.html), different types of Polymers only for $2.1/Kg *(less than a coffee cup)*, and make lots of these with a very fast rate and low cost

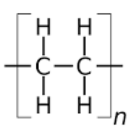
Polymer = Poly (many) + mer (structure)



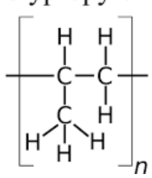




Polyethylene



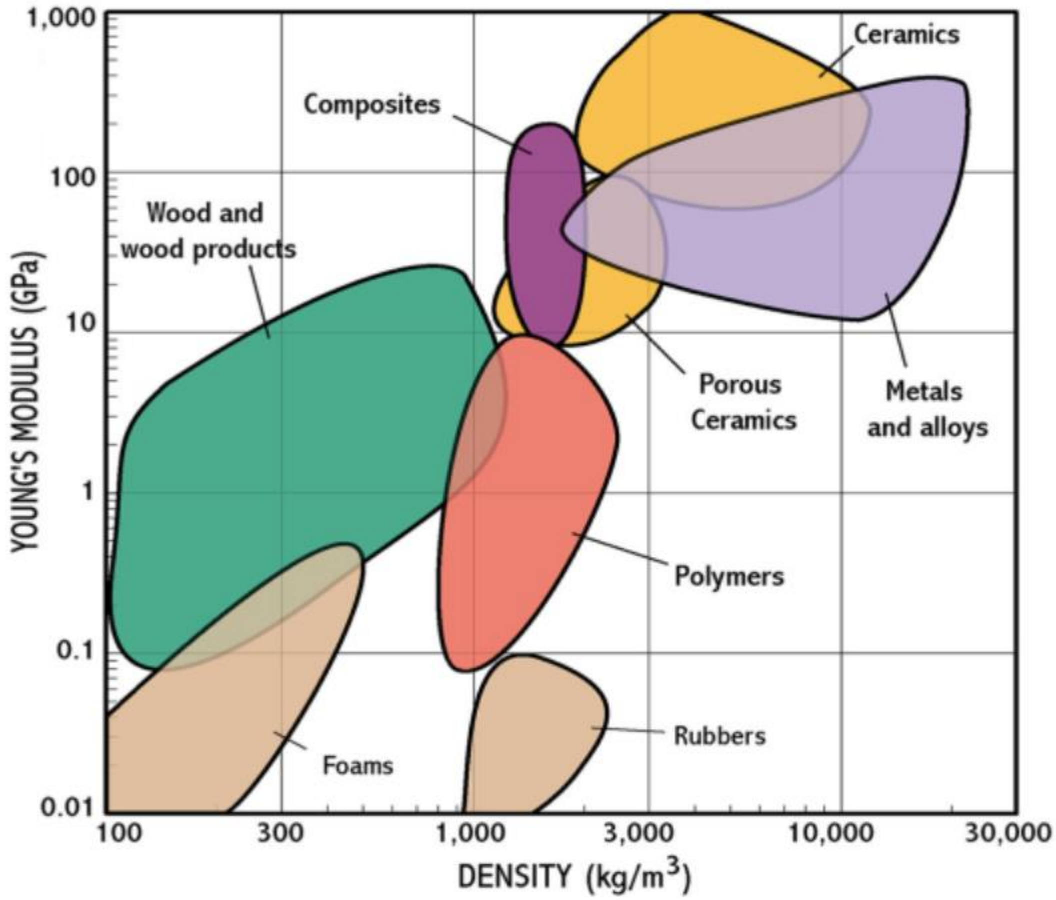
Polypropylene



PET: Polyethylene Terephthalate ---- use for for carbonated beverages, thin wall but capable of containing gaseous carbonated beverages without exploding even if they are dropped.

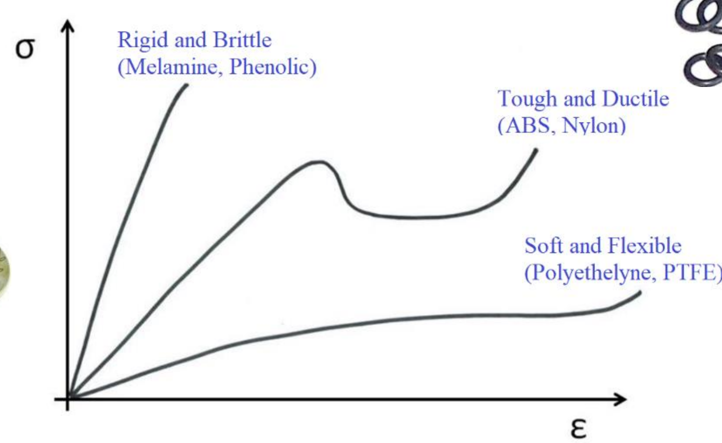
Interested in knowing how much pressure a coca bottle can take?

Where are Polymers in families of materials (Elasticity - Density)



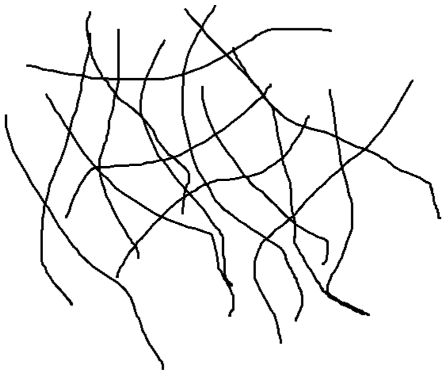
(Ashby, Materials selection in mechanical design)

Stress-Strain Behaviour of Plastics:

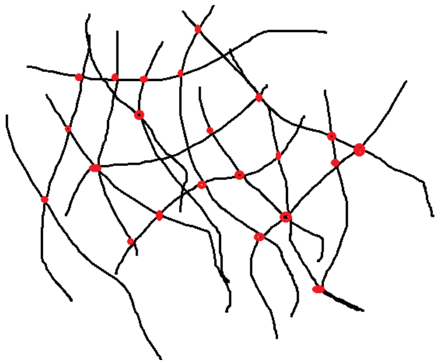


Polymer network structure:

Thermoplastic: Linear



Thermoset: Cross-linked



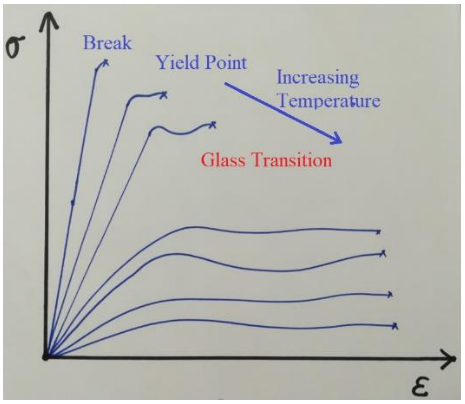
Thermoplastic*:* Thermoplastics pellets soften when heated and become more fluid as additional heat is applied. The curing process is completely reversible as no chemical bonding takes place. This characteristic allows thermoplastics to be remolded and recycled without negatively affecting the material's physical properties; It is high-impact resistance, remolding/reshaping capabilities, Chemical resistant, but Can melt if heated.

But most likely you don't want your kitchen spatula, or the pot handle starts to melt while you are cooking!!!

Thermoset*:* Thermoset plastics contain polymers that cross-link together during the curing process to form an irreversible chemical bond. The cross-linking process eliminates the risk of the product re-melting when heat is applied, making thermosets ideal for high-heat applications such as electronics and appliances. It has high levels of dimensional stability; but Cannot be recycled

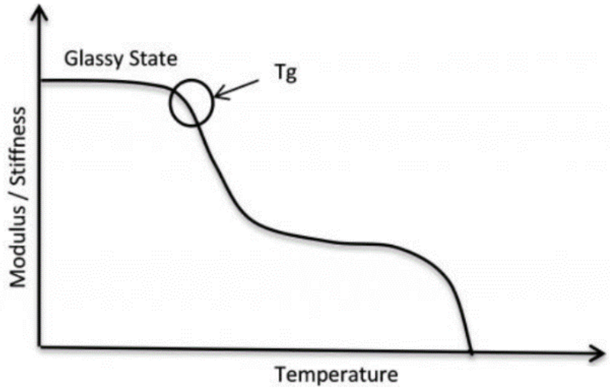
Mechanical Properties of Polymers

The mechanical properties of Polymers changes under heat and pressure. As the temperature increases the breaking point decreases, and the material becomes formable

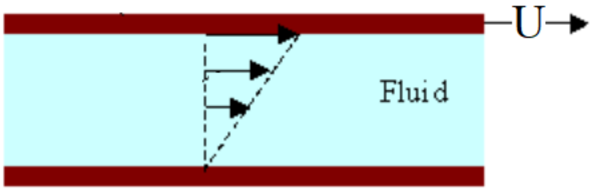


Glass Transition:

The glass–liquid transition, or glass transition, is the gradual and reversible transition in amorphous materials, from a hard and relatively brittle "glassy" state into a viscous or rubbery state as the temperature is increased.



Viscosity: the resistance to shear



$$\tau = \mu \frac{\partial U}{\partial y}$$

You can compare the viscosity of a molten plastic about 10 times the viscosity of honey.

No	Material	Viscosity (Pa.Sec)
1	Water	0.001
2	Olive oil	0.1
3	Honey	10
4	Liquid Thermoplastic	100

The injection molding machine:

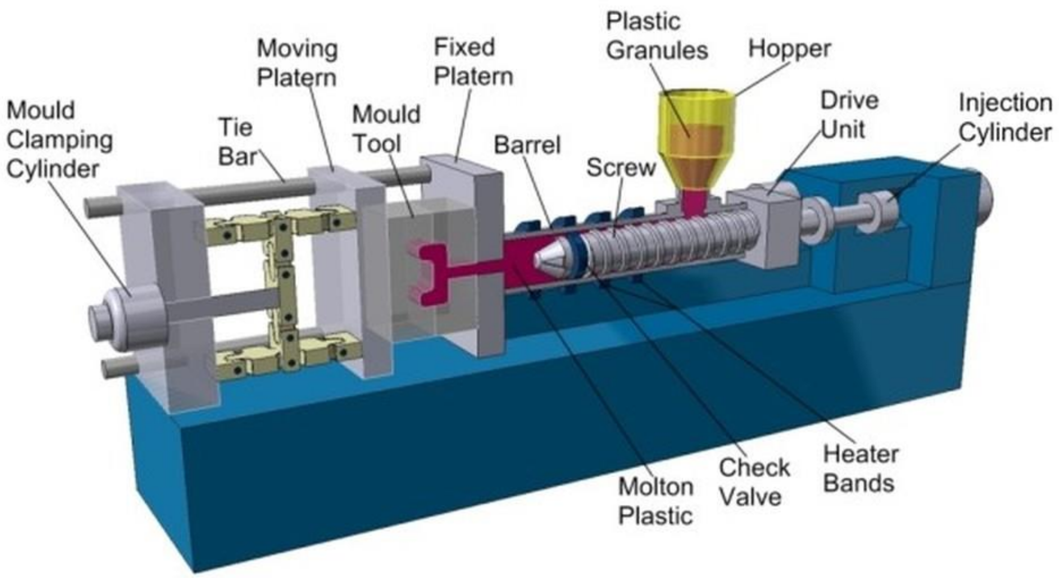
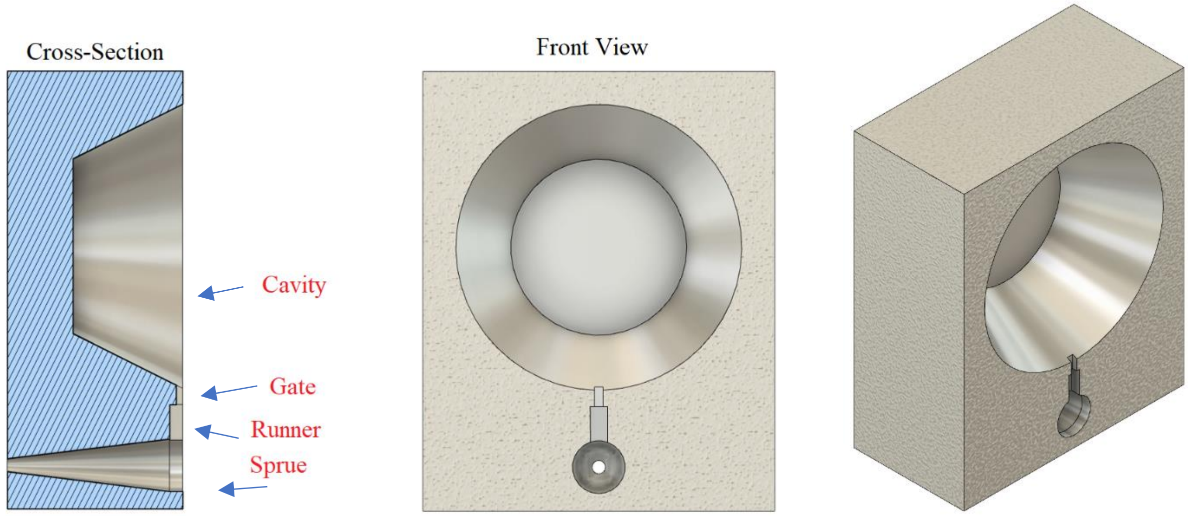


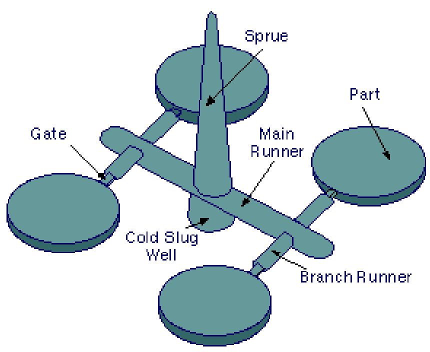
Image source: https://predictabledesigns.com/introduction-to-injection-molding/

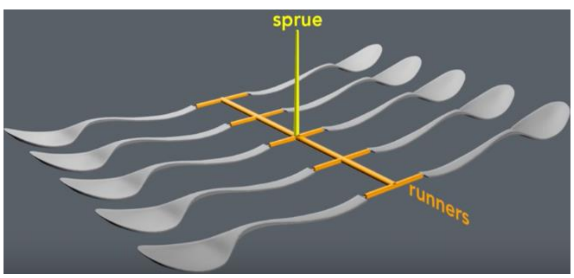
Injection molding machines incorporate a large screw to force molten plastic into the mold at high pressure. An injection mold consists of two halves that are clamped together to form a cavity in the shape of the part to be manufactured. Molten plastic is then injected at high pressure into this cavity. The high pressure ensures that the plastic fills in every corner and narrow spots of the mold cavity. The plastic needs time to cool. After cooling, the two halves of the mold are pulled apart, and the part is ejected. Although the cost of the molds themselves are incredibly expensive, no technology can beat injection molding when it comes to producing millions of identical copies of a part at an incredibly speed and low price.

The features of the injection mold tool:



Molten plastic first flows out of the extruder through the sprue. From there, the plastic travels down a wide channel called a runner, before a constriction known as the gate. This is the last opening that leads to the mold cavity.





Looking at an injection molding part, the gate mark on the part indicates where the sprue was attached, and thus at which side of the mold the plastic was injected from. Ejector pin divots, usually on the opposite side of the part from the gate mark, indicate where the part was pushed out of the cavity. Finally, parting lines will occur where the mold splits, and where side pulls make a seam with the mold halves.

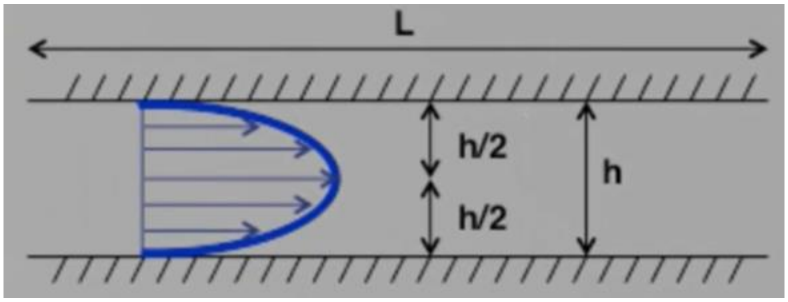
Engineering analysis of Injection Molding:

There are several process parameters which need to be determined depends on the size and geometry of a part:

- **Clamping force** (to hold the two halves of the mold)
- **Injection Pressure** (necessary pressure to force the plastic into mold cavity)
- **Shot size** (the volume of the plastic, which is part volume, runner, sprue, and gate volume)
- **Cooling time**

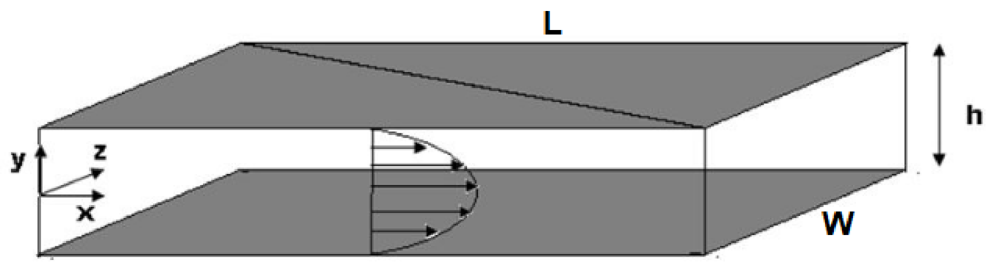
Image source: Melody Reports, Global injection molding market strategies,

Assume flow is steady, laminar, and constant. The Navier-Stoke equations relate the pressure gradian in x to the velocity in y direction as:



$$\frac{dP}{dx} = \mu \frac{d^2U}{dy^2}$$

Assume a 2D rectangular channel:



Assume that the volume flow rate Q of plastic into the mold cavity. Solving the Navier-Stoke equations, the pressure drop across the cavity ΔP can be estimated as:

$\Delta P = \frac{12\mu QL}{Wh^3}$

The time to fill the mold can be calculated as: $t = \frac{LWh}{Q}$

From these two equations, the required time to fill in the rectangular mold is $t = \frac{12\mu}{\Delta P}(\frac{L}{h})^2$

And the clamping force, can be estimated as $F = \frac{1}{2}\Delta P \times WL$

Example 1) A molten plastic (Polypropylene) flow into a rectangular box of L= 100mm W=50mm h=2.5mm, if the pressure drop in the mold is about $\Delta P = 3MPa$. Estimate the time it takes to fill in the mold? Assume the viscosity of $\mu = 85\ Pa.Sec$

Answer=0.544 sec

Example 2) If the gage pressure at the inlet of an injection mold cavity is 15 MPa, and the cavity has a length of 0.25 m, and a width of 0.04 m, estimate the average clamping force required.

Answer=75000N

Shot Size:

The shot size of an injection molding machine comprises the volume of the following features: Part cavities, Gates, Runners, Spruce. The shot size consists of the plastic injected out of the screw and into the mold. Since the mold houses the sprue (ejected with the part), channels, gates, and the part itself; the shot needs to carry enough plastic to fill all these portions of the mold. The hopper holds the unmelted pellets, and the screw melts the pellets and holds the shot until injection.

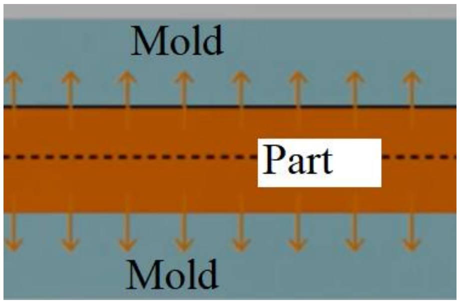
Polymer changes volume with pressure and temperature. Insufficient packing of material inside the mold results in shrinkage and dishing (with enough material, there should be only 1% shrinkage)

Imagine a sponge which tries to shrink but it is glued to the wall of a container, it results in residual stresses.

Residual stresses in injection mold parts arise primarily due to uneven cooling. For example, if thick walls are adjoined to thin walls in an injection molded part, the difference in time required to cool each segment leads to a difference in shrinkage rate throughout the part. This causes warpage, and if the tendency to warp is restricted, internal (residual) stresses result. It results in a defect.

Cooling time:

The temperature of molten plastic reduces gradually. The cooling time is defined as the time until the centerline of the part to reach ejection temperature. At the end of the cooling time the part should be fully solidified, meaning that even the centerline of the piece needs to be at the desired ejection temperature.



we can write a differential equation that says, locally, the change in temperature of the part by time is equal to:

$$\frac{\partial T}{\partial t} = \frac{k}{\rho C_p} \frac{\partial^2 T}{\partial y^2}$$

K is the thermal conductivity, ρ is the density, and C_p is the specific heat. The ratio K over $\rho C_p$is called α. And α is known as the thermal diffusivity, which is a measure of the velocity of heat flow through the material. So, we can rewrite the equation as:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial y^2}$$

If we solve this equation for the following boundary conditions:

T_m : molten plastic temperature

T_w : wall of the mold temperature

T_e : ejection temperature

$$t_{cool} = \frac{h^2}{10\alpha} Ln(\frac{4}{\pi} \times \frac{T_m - T_w}{T_e - T_w})$$

Example 3) A molten plastic (Polypropylene) into a rectangular box L= 100mm W=50mm h=2.5mm if the pressure drop across the mold is about $\Delta P = 3MPa$. Estimate the cooling time?

$T_m = 250C$, $T_w = 75C$, $T_e = 125C$, $C_p = 1920\,\frac{J}{Kg.C}$, K=0.6 W/m.C, $\rho = 945\text{kg/m}^3$

Answer=2.92 Sec

------------**Side notes** --

Specific heat: *Specific heat or heat capacity or thermal capacity is a physical property of matter, defined as the amount of heat to be supplied to a given mass of a material to produce a unit change in its temperature. The SI unit of heat capacity is joule per kelvin. The specific heat of thermoplastic is in the range of 2000* $\frac{J}{Kg.C}$.

Thermal conductivity*: The quantity of heat that passes in unit time through unit area of a substance whose thickness is unity, when its opposite faces differ in temperature by one degree. The SI derived unit of thermal conductivity is watt per metre kelvin (W/m.K) For most of thermoplastics, the thermal conductivity at 25°C falls in the range of 0.1 W/m.K (for polypropylene) and 0.5 W/mK (for high density polyethylene)*

Density: *Density of polypropylene is in a range of 950 kg/m*^{3}

Defect in Injection Molding Parts:

There are several defects type in injection molding parts. Flash results from either insufficient clamping force or too high of injection or packing pressure. Sink marks result from the shrinking of plastic in thicker regions of an injection molded part. A short shot is a consequence of either an insufficient volume of plastic begins injected into the mold, or premature freezing of the gates or sprue. Burning marks on the final part. The main reasons for these defects can be:

Void: Too small a shot size, not enough pressure to fill mold, and too low a temperature may result in small voids in the part.

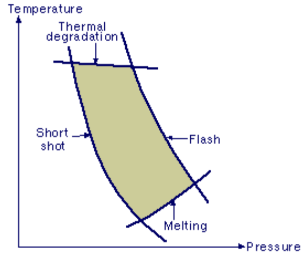
Flash: Flash is a defect where excessive material is found at locations where the mold separates, notably the parting surface, movable core, vents, or venting ejector pins. The main reasons might be High pressure and High temperature, Low clamp force, Gap within the mold

Burning degradation: Burn marks are small, dark or black spots on the part surface. This phenomenon is also often referred to as dark streaks or specks. Air trapped in pockets may compress, heat up and cause burn marks.

Material degradation: This can be caused by excessive injection speed, residence time or melt temperature. Improper screw or runner system design may also lead to material degradation.

The process window:

We have control only over Pressure, and Temperature; depends on the material, we must adjust P and T so that the parameters are inside the process window *(more discussion in class).*



To study more, please read the followings:

1- **Top-10 Injection Molding Defects And How To Fix Them Insert Molding**

2- **Molding Defects, Causes & Corrective Actions**

Insert molding:

A typical application of insert molding is to include one or more threaded metal inserts in a plastic part when that part is intended to mate to another part in an assembly. Plastics alone may not have enough mechanical properties to withstand the forces required to fasten two parts together. For example, threads in a plastic part can become worn over repeated usage which can result in a failed part. Metal inserts help reinforce the properties of the plastic and ensure reliable fastening over repeated use of the part. This combination of plastic and metal allows designers to take advantage of the weight reduction of plastics and the strength of metal.

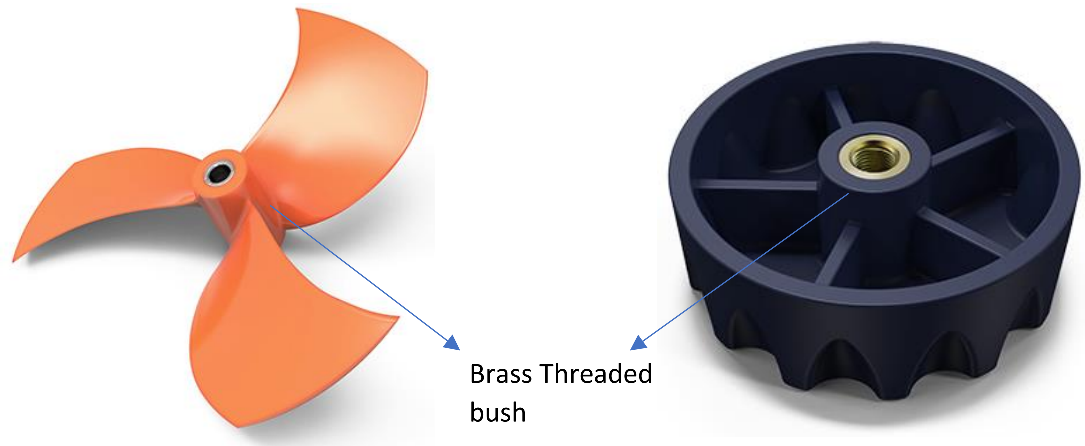
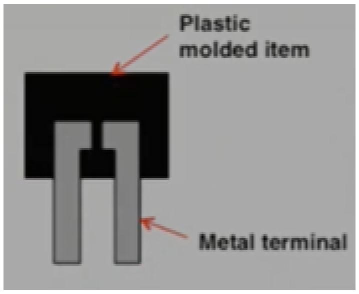


Image Source: https://www.manufacturingtomorrow.com



Over molding:

Overmolding is an injection molding process that allows an additional different layer of plastic to be added to an existing molded part to provide a combination of characteristics that a single material cannot provide. The most common applications is to add a soft, functional, hand-friendly layer of rubber-like material, typically a TPE (thermoplastic elastomer) over a hard substrate. Overmolded materials can be found on anything from medical devices and hand-tools to toothbrushes. *(For more reading on this topic please check https://www.protolabs.com/)*

Metal Injection Molding:

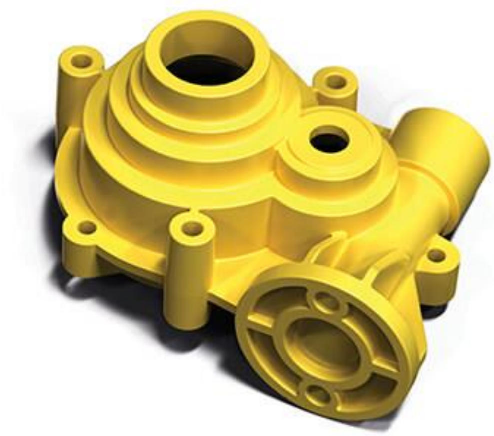
Metal injection molding consists of injecting a metal powder and polymer and wax binder material into a mold using a plastic injection molding machine. After the part is ejected, some of the binder is removed and the part is sintered to complete the metal part.

Image source: http://www.gc-manufactures.com/machining/steel-injection-molding.html

The process involves combining fine metal powders with plastic binders which allow the metal to be injected into a mold using equipment like standard plastic injection molding machines. After the part is molded and before the binders are removed, the part is referred to as a "green part". The next step is to remove the binders with solvents and thermal processes. The resultant metal part is sintered at temperatures great enough to bind the particles but not melt the metal.

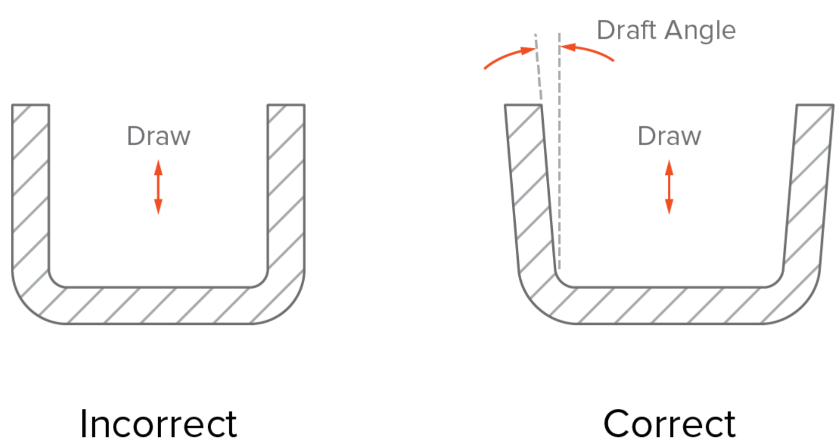
Design for Moldability:

Designers and engineers should consider different factors if their parts will be injection molded from wall thickness, radii to ramps and ribs. It is important to design injected molded parts to have an uniform thickness, So that the plastic cools uniformly.



Draft angle:

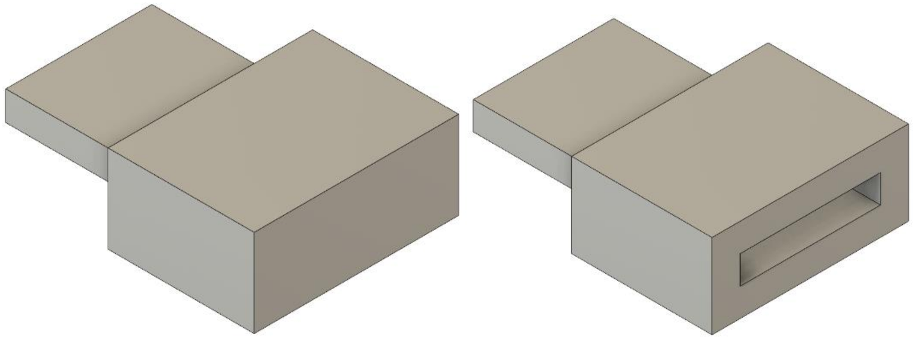
Adding a draft angle to the faces of your part is helpful to release it from the tool.

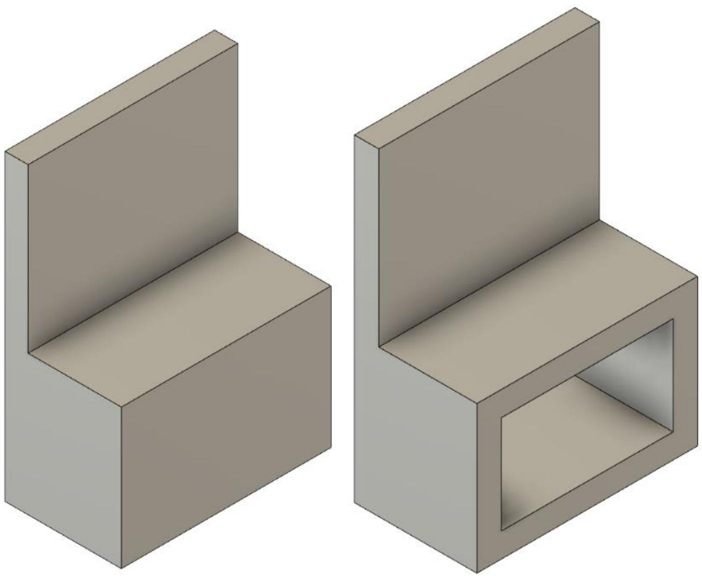


The purpose of a draft angle is to aid in part ejection and prevent the solidified plastic adhesion to the mold sidewalls from creating to much shear force that the piston, or the rest of the part would have to overcome when pushed out of the mold. Recommend minimum draft angles are <u>one degree</u> on untextured core and at least <u>three degrees</u> on textured cavity surfaces.

Wall Thickness

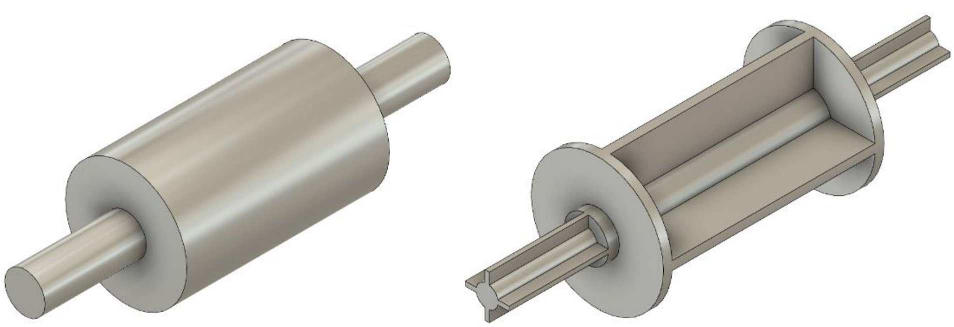
Consistent wall thickness minimizes the potential for warped or distorted parts. Try to core out parts to eliminate thick wall.





Core-cavity

Core out parts to eliminate thick walls:

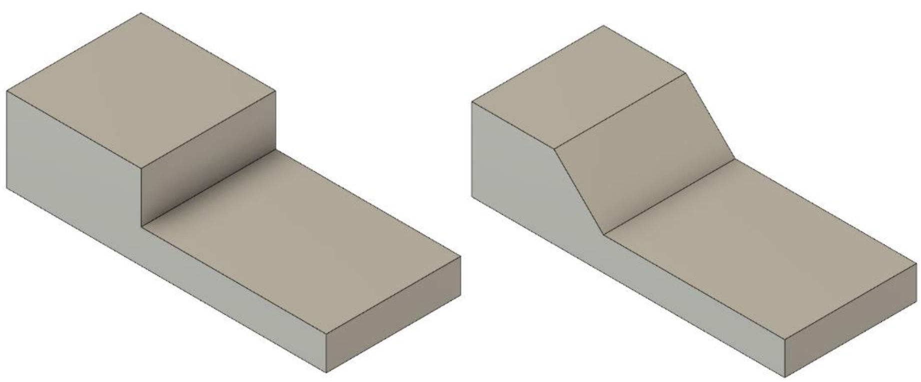


Wall Thickness Guidelines (*Reference: Portolab Manufacturing***)**

No.	Material	Recommended Wall Thickness
1	ABS	0.045 in. - 0.140 in.
2	Acetal	0.030 in. - 0.120 in.
3	Acrylic	0.025 in. - 0.500 in.
4	Liquid Crystal Polymer	0.030 in. - 0.120 in.
5	Nylon	0.030 in. - 0.115 in.
6	Polycarbonate	0.040 in. - 0.150 in.
7	Polyester	0.025 in. - 0.125 in.
8	Polyethylene	0.030 in. - 0.200 in.
9	Polyphenylene Sulfide	0.020 in. - 0.180 in.
10	Polypropylene	0.035 in. - 0.150 in.
11	Polystyrene	0.035 in. - 0.150 in.
12	Polyurethane	0.080 in. - 0.750 in.

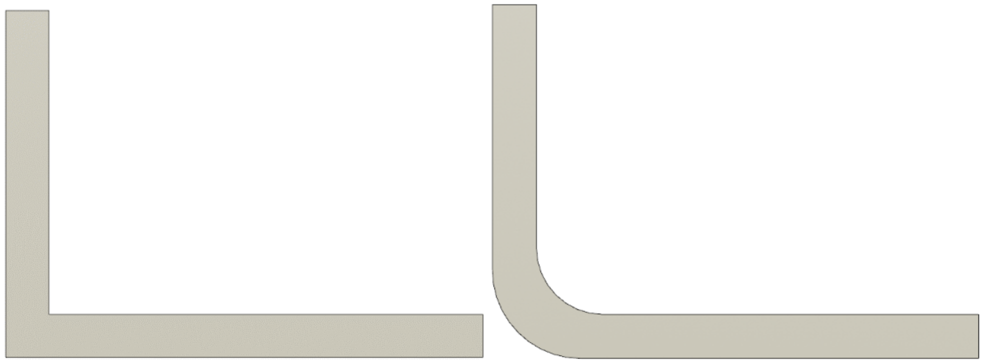
Ramps

Eliminate sharp transitions that cause molded-in stress.



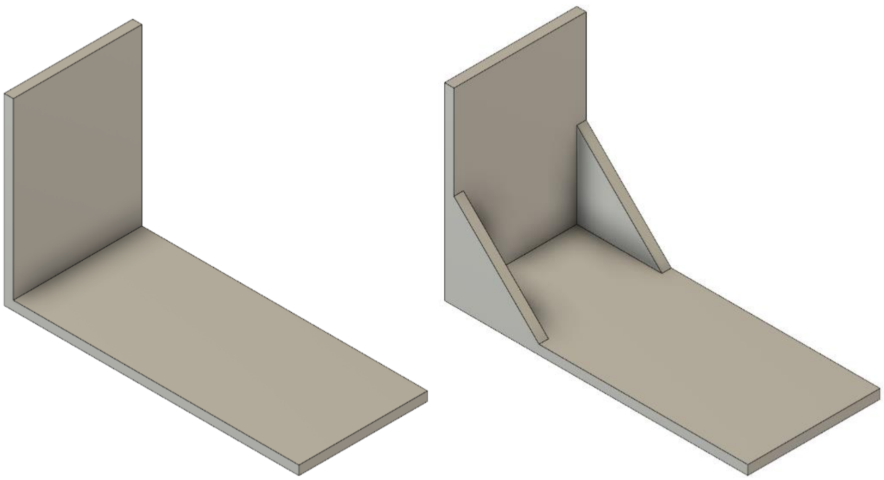
Fillet

Sharp corners weaken parts, causing molded-in stress from resin flow.



Support:

Design part features that support themselves:



Tolerances

Shrink tolerance varies from 0.002 in./in to 0.025 in./in.

Material Selection

No.	Material	Property
1	**Polypropylene**	• Soft • Tough • Cheap • Chemical resistant
2	**Polyethylene**	• Soft • Tough • Cheap • Chemical resistant • Low Density
3	**Polystyrene**	• Hard • Clear • Cheap • Brittle but can be toughened
4	**ABS**	• Inexpensive • Impact Resistant • Equipment and handheld housings
5	**Acetal**	• More expensive • Strong • Good lubricity and machinability • Very sensitive to excess wall thickness
6	**Liquid-crystal polymer**	• Very expensive • Very strong • Fills very thin parts • Weak knit lines
7	**Nylon**	• Reasonable cost • Very strong • Susceptible to shrink and warp • Absorbs water - dimensional and property change
8	**Polycarbonate**	• Moderate cost • Very tough • Good dimensional accuracy • Susceptible to chemical stress cracking, voids