Injection Molding Principles
Plastics

Advantages

Light Weight
Complex Parts - Net Shape
Variety of Colors (or Clear)
Corrosion Resistant
Electrical Insulation
Thermal Insulation
High Damping Coefficient

Disadvantages

Creep
Thermally Unstable- Can’t withstand Extreme Heat
U-V Light Sensitive
Difficult to Repair/Rework
Difficult to Sort/Recycle
Two basic types of plastics
- Thermoset: Heat hardening, undergoes chemical change
- Thermoplastic: Heat softening, undergoes physical change

A wide variety of plastic manufacturing processes exist
- Extrusion
- Thermal Forming
- Foaming
- Casting
- Molding
- Transfer Molding
- Blow Molding
- Injection Molding
- Expansion
- Lamination (Calendering)
- Spinning
- Solid-Phase Forming
- Compression Molding
- Rotational Molding
- Reaction Injection Molding
Injection Molding

Basic Process--
Heat plastic to beyond Glass Transition Point
Close mold, creating a fully enclosed cavity
Pressurize plastic melt and inject into mold cavity
Allow plastic melt to solidify, while keeping mold filled
Open mold and eject part

Key issues for design of part and mold

1) Filling Mold & Holding Pressure
   Mold layout designed to enhance fill (use good fluid flow principles)
   After part is filled, packing pressure is maintained, so part will not shrink
   away from walls as it solidifies

2) Ejecting Part
   Part should hang on moving side as it retracts (pulling free of fixed side)
   Ejector pins then push part out of moving side of mold
   Taper or draft required to ensure ejection
Plastic flows from injection nozzle into sprue then into the runners and finally through the gate into the part. Want to balance the mold so that all cavities fill at the same time.
Material is stored (in pellet form) in the hopper
Band heaters heat material as it moves through shooting pot
Stroke of the plunger meters the shot size
Virtually ALL industrial presses are screw type presses

Added benefits of screw

1) Larger throughput
2) Obtain a more homogeneous melt (better mix)
3) More consistent from shot to shot
4) Added heat to melt- from action of screw
Press Parameters

3 parameters commonly used to describe press capacity

1) **Clamping force** - Force available to hold platens together (tons)
   Can be from Hydraulic/Pneumatic Cylinder
   Mechanical Toggle Clamp

2) **Shot size** - Amount of material that can be transferred to mold in a shot (given in either cm³ or ozs)

3) **Injection Pressure** - Maximum pressure that can be developed at the sprue to force the plastic into the mold cavity
Injection Molding Defects

**Short Shot** - Insufficient material put in cavity / material solidifies too soon

**Ejector Pin Marks** - Ejector pins not flush / undersized or plastic too soft at ejection

**Weld Lines** - Occur wherever flow paths come together

**Sink Marks** - Material shrinks away from walls of mold cavity

**Residual Stresses** - Generated by constraining part while cooling

**Parting Lines** - Plastic seeps through seam between mold sections

**Jetting** - Melt moving rapidly, cools unevenly and traps flow lines

**Flashing** - Material overflows cavity (too much material, not enough clamp force)
Short Shot

Unfilled Section of Mold

Unfilled Sections of Mold
Flash
Plastic Pushes into Area Between Mating Surfaces

Part w/ Moderate-Heavy Flash
Weld lines are created when two flow fronts come together in the mold. Weld lines cause a decrease in the strength of the part. The cooler the fronts are when they meet, the more pronounced the weld lines will be. 

Can result from poor gate placement

Unavoidable w/ Solid Cores
Jetting

Jetting occurs at high fill rates when there is a large open space between the gate and the opposite wall. Material stream shoots to the opposite wall and freezes. Stream of fresh material then folds over this and freezes, trapping flow lines of material. Air can often be trapped between folds.

To reduce the chance of jetting, gates should be located so that entering material flows into wall, not into open section.
Ejector Pin Marks

Marks are often left on the part in the area where the ejector pins were. These marks have three possible causes:

1) Pin above flush
2) Pin below flush
3) Clearance around pin

To some degree, these marks are unavoidable, thus one should try to place the ejector pins on hidden areas of the part. Another defect occurs when one tries to eject the part before it solidifies and pins push through the part.
Sink Marks

Sink marks occur at excessively thick wall sections, or where there are abrupt changes in thickness—thick sections solidify too late and shrink away from the wall.

Proper design eliminates unneeded material (ribs, core out sections).

**Bad Design**

**Improved Design**

When unavoidable, sink marks can be masked by surface texture.
THE GATEWAY COALITION

Runner Design

1) Keep runners as short as possible

2) Use a cold well at the end of each branch- collect solid chunks

3) Use cross sections that minimize perimeter for a given area
   Circles are best followed by rounded trapezoids and then trapezoids

4) Use good flow principles (round corners, etc)

5) Size the (effective) diameters of runners based on length
   Determine total flow length from sprue to gate
   Use table to determine needed diameter for this length- this value is minimum diameter at END of runner
   Increase the diameter of the upstream runner by 20% at each 90° turn
   When n multiple runners branch off a main feed runner, the diameter of the main runner should be increased by n*20%
### Runner Sizing

<table>
<thead>
<tr>
<th>Runner Length (in)</th>
<th>3</th>
<th>6</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Material</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ABS</td>
<td>0.093</td>
<td>0.109</td>
<td>0.156</td>
</tr>
<tr>
<td>Acetal</td>
<td>0.062</td>
<td>0.093</td>
<td>0.125</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.125</td>
<td>0.156</td>
<td>0.187</td>
</tr>
<tr>
<td>Cellulose acetate</td>
<td>0.093</td>
<td>0.109</td>
<td>0.156</td>
</tr>
<tr>
<td>Cellulose acetate butyrate</td>
<td>0.093</td>
<td>0.109</td>
<td>0.125</td>
</tr>
<tr>
<td>Ionomer</td>
<td>0.062</td>
<td>0.093</td>
<td>0.125</td>
</tr>
<tr>
<td>Nylon 6/6</td>
<td>0.062</td>
<td>0.078</td>
<td>0.093</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>0.125</td>
<td>0.156</td>
<td>0.203</td>
</tr>
<tr>
<td>Polyethylene</td>
<td>0.062</td>
<td>0.093</td>
<td>0.125</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>0.062</td>
<td>0.093</td>
<td>0.125</td>
</tr>
<tr>
<td>Polyphenylene oxide</td>
<td>0.125</td>
<td>0.156</td>
<td>0.203</td>
</tr>
<tr>
<td>Polyphen. sulfide</td>
<td>0.125</td>
<td>0.156</td>
<td>0.203</td>
</tr>
<tr>
<td>Polysulfone</td>
<td>0.156</td>
<td>0.187</td>
<td>0.218</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>0.093</td>
<td>0.109</td>
<td>0.125</td>
</tr>
<tr>
<td>Rigid PVC</td>
<td>0.125</td>
<td>0.187</td>
<td>0.250</td>
</tr>
</tbody>
</table>

#### Example with ABS

**Length = 3"**

- Diameter = 0.093"

**Length = 6"**

- Diameter = 0.153" (1.4*0.109")

---

Table 2 “Secrets of gate, runner, and vent design,” Douglas M. Bryce. *Plastics Design Forum.* Sept 94
Gate Design

1) Gate so that material flow hits a nearby wall
2) Gate into the thick sections - so material flows thick to thin
3) Gate into an understressed region of the part
   (Gates introduce residual stresses as they freeze before other sections)
4) “Hide” gates whenever possible
5) Size gates based on (maximum) wall thickness of part
   Also depends on how well the material flows (viscosity)
   Determine relative material viscosities by comparing the minimum allowable wall thicknesses for different materials
**Gate Sizing**

<table>
<thead>
<tr>
<th>Material</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS</td>
<td>0.030</td>
<td>0.125</td>
</tr>
<tr>
<td>Acetal</td>
<td>0.015</td>
<td>0.125</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.025</td>
<td>0.250</td>
</tr>
<tr>
<td>Nylon, amorphous</td>
<td>0.030</td>
<td>0.125</td>
</tr>
<tr>
<td>Nylon, crystalline</td>
<td>0.015</td>
<td>0.125</td>
</tr>
<tr>
<td>Phenolic</td>
<td>0.045</td>
<td>1.000</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>0.040</td>
<td>0.400</td>
</tr>
<tr>
<td>Polyester (thermoplastic)</td>
<td>0.025</td>
<td>0.125</td>
</tr>
<tr>
<td>Polyester (thermoset)</td>
<td>0.040</td>
<td>0.500</td>
</tr>
<tr>
<td>Polyethylene (hd)</td>
<td>0.020</td>
<td>0.250</td>
</tr>
<tr>
<td>Polyethylene (ld)</td>
<td>0.030</td>
<td>0.250</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>0.025</td>
<td>0.300</td>
</tr>
<tr>
<td>Polyphenylene oxide, modified</td>
<td>0.030</td>
<td>0.400</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>0.030</td>
<td>0.250</td>
</tr>
<tr>
<td>PVC</td>
<td>0.040</td>
<td>0.400</td>
</tr>
</tbody>
</table>

\[ t = \text{(wall) thickness of part} \]
\[ d \text{ (depth)} : \quad 0.4 \cdot t \leq d \leq 0.8 \cdot t \]
\[ w \text{ (width)} : \quad 2 \cdot d \leq w \leq 3 \cdot d \]
\[ l \text{ (land)} : \quad l \leq 0.030" , \quad l \approx 0.5 \cdot d \]

Vents allow air to escape ahead of the plastic melt
Vents help reduce--
Injection pressure and clamping force
Cycle time
Warping & shrinking
Residual stresses
**Vent Sizing and Frequency**

Depth ($d$) from table

- Width: $w \geq 0.125''$
- Land: $0.030'' \leq l \leq 0.125''$

Note that runners should be vented if possible

**Recommended Vent Depth (in)**

<table>
<thead>
<tr>
<th>Material</th>
<th>Cavity</th>
<th>Runner</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS</td>
<td>0.002</td>
<td>0.004</td>
</tr>
<tr>
<td>Acetal</td>
<td>0.0007</td>
<td>0.0015</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.002</td>
<td>0.004</td>
</tr>
<tr>
<td>Cellulose acetate</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Cellulose acetate butyrate</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Ionomer</td>
<td>0.0007</td>
<td>0.0015</td>
</tr>
<tr>
<td>Nylon 6/6</td>
<td>0.0005</td>
<td>0.001</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>0.002</td>
<td>0.004</td>
</tr>
<tr>
<td>Polyethylene</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Polyphenylene oxide</td>
<td>0.002</td>
<td>0.004</td>
</tr>
<tr>
<td>Polysulfide</td>
<td>0.0005</td>
<td>0.001</td>
</tr>
<tr>
<td>Polysulfone</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>0.001</td>
<td>0.002</td>
</tr>
<tr>
<td>Rigid PVC</td>
<td>0.002</td>
<td>0.004</td>
</tr>
</tbody>
</table>

Vent 30% of runner length on both sides.

Vent 20% of cavity perimeter if runners are vented. By vent 20% of the cavity means that if one sums the width of each vent and divide the result by the total perimeter of the cavity, the ratio should be around 20%.

Vent 30% of cavity otherwise

The part shown on the right is to be injection molded. 16 pieces should be made in each shot. (Material is polycarbonate). Design the mold.

1) Layout
The configuration shown on the right keeps the runners small, with few bends, and shoots the melt into the mold wall. One weld line will be present on each part, but this is unavoidable.
Example- Runner Design

2) Runner Size

Total Runner Length = 1.2” + 7.5”/2 + 3.8” = 8.75”

From table EFFECTIVE diameter is 0.203.”

For a flat trapezoid, with an angle of 15°, and height and width chosen to minimize the perimeter for a given area...

\[
h = \left( \frac{d}{2} \right) * \sqrt{\pi} * \cos(15^\circ)
\]

\[h = 0.871 * d\]

\[w = \left( \frac{d}{2} \right) * \sqrt{\frac{\pi}{\cos(15^\circ)}} \times [1 + \sin(15^\circ)]\]

\[w = 1.135 * d\]

\[h = 0.871 * (0.203”) = 0.176”\]

\[w = 1.135 * (0.203”) = 0.230”\]
Example- Upstream Runners

Recall that these dimensions (effective diameter = 0.203") are for the runners nearest the gate.

Four of these cluster runners (n=4) branch off of each feeder. Effective diameter=(1+n*0.2)*d_{branch}
\[d_{feed} = (1+4\times0.2)\times(0.203") = 0.366"
\[h=0.871\times(0.366") = 0.319"
\[w=1.135\times(0.366") = 0.415"

Two of these feeder runners (n=2) branch off of the main runners.
\[d_{main} = (1+2\times0.2)\times(0.366") = 0.512"
\[h=0.871\times(0.512") = 0.446"
\[w=1.135\times(0.512") = 0.581"
Example- Gates

Wall thickness is given as 0.100” - t=0.100.”
Gate depth : \(0.4 \times t \leq d \leq 0.8 \times t\)

From tables, the minimum wall thickness when injection molding Polycarbonate is 0.040,” which is comparatively high. This indicates that Polycarbonate does not fill or flow as well as most plastics.
Select \(d=0.7\times t=0.7\times(0.100”) = 0.070”\)
Gate Width : \(2 \times d \leq w \leq 3 \times d\)
Select \(w=2.75\times d=2.75\times(0.070”) = 0.193”\)
Gate land : \(l \leq 0.030,” \quad l \approx 0.5 \times d\)
Select \(l=0.030”\)
Example- Vents

Depth of vents determined from table
- Mold cavity vent depth $d_{cv}=0.002”$
- Runner vent depth $d_{rv}=0.004”$

Vent width: $w_v \geq 0.125”$
  - Choose $w_v=0.125.”$

Vent land: $0.030” \leq l_v \leq 0.125”$
  - Choose $l_v=0.045.”$

Vent 30% of runner length
  - Spacing between vents on runners is $0.42”$

Vent 20% of mold cavity
  - Spacing between vents in cavity is $0.500”$
Estimating Clamping Force

Clamping force is needed to overcome the force exerted by the pressure of the plastic melt against the mold cavity. A reasonable estimate of clamping force may be obtained by multiplying the pressure by the area of the mold in the parting plane. Assume that 1/2 of the pressure drops uniformly through the runner system. Assume that the remaining pressure holds constant in the mold cavity. (Note that these assumptions are very conservative.)

\[ F = 0.75 \times P_i \times \sum (l_r \cdot w_r) + 0.5 \times P_i \times \sum A_{mc} \]

- \( P_i \) = Injection Pressure
- \( A_{mc} \) = Area(s) of mold cavities (in parting plane)
- \( l_r \) = Length of runners
- \( w_r \) = Width of runners

**Typical Injection Pressures (kpsi)**
- Polycarbonate (Lexan) 16
- Phenylene Oxides (Noryl) 14
- Polyester (Valox) 10
- ABS (Cycolac) 10

(From GE Product Guides)
Example - Clamping Force

Force for previous mold

1) Mold Area
   Treat parts as perfect squares
   \[ A = 2'' \times 2'' - 1.8'' \times 1.8'' \]
   \[ A = 0.76 \text{ in}^2 \]
   \[ \Sigma A = 16 \times 0.76 \text{ in}^2 = 12.2 \text{ in}^2 \]

2) Runner Area
   Main: \( (7.5'') \times (0.581'') = 4.4 \text{ in}^2 \)
   Feed: \( 4 \times (3.8'') \times (0.415'') = 6.3 \text{ in}^2 \)
   Branch: \( 16 \times (1.2'') \times (0.230'') = 4.4 \text{ in}^2 \)
   \[ \Sigma A = (4.4+6.3+4.4) \text{ in}^2 = 15.1 \text{ in}^2 \]

3) Typical Polycarbonate Injection Pressure: 16000 psi

4) Clamping force
   \[ F = 0.75 \times (16000 \text{ psi}) \times (15.1 \text{ in}^2 ) + 0.5 \times (16000 \text{ psi}) \times (12.2 \text{ in}^2 ) \]
   \[ F = 280000 \text{ lbs} = 140 \text{ tons} \]
THE GATEWAY COALITION

Project 3

To injection mold a Drexel keychain in Hess Lab.
1. Select plastic material for the keychain (read Chapter 10-5 to 10-8)
2. Select color for the keychain (red, blue and yellow)
3. Make 50 good quality keychains by adjust pressure, temperature, time, etc. on the machine
4. Logo design for the keychain
5. Prepare 10 min. power Point presentation and detail report for this project
Available Plastic Materials in Hess Lab

1. Nylon Resin (Zytel)
2. Polyethylene (PE)
3. Acetal Resin (Delrin)
4. Polycarbonate Resin
5. Polyvinyl chloride, PVC
6. Polypropylene
7. Polyetherimide
8. Elastomers