**Injection Molding Outline**

- basic process description
  - physical processes involved
  - tight tolerances on parts
  - high pressures
- machine description
  - terminology of machine and parts
  - functional components on the machine
    - resin feed system (hopper / dryer)
    - screw, barrel and accumulator
    - temperature control
    - mold (cavity, runners, cooling channels, ejector system)
    - clamping system
    - unit to supply power (hydraulic)
    - control unit
- description of the cycle
- part cooling
  - models for cooling
    - penetration model
    - cooling of slab
    - cooling parameters
- multicavity molds
- areas of concern
  - part filling
  - part cooling
  - defects
- comparison of injection vs extrusion
The Injection Molding Hardware

Basic Description of Injection Molding

- need a means of melting the plastic
  (plasticating unit)
- need a means of shaping the part
  (a mold has a cavity cut in the shape of the part)
- need to cool the part
  (cooling channels in the mold)
- need to transport the melt from the extruder to the mold
  (flow channels called runners)

This is a cyclic process

<table>
<thead>
<tr>
<th>Step</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>melting</td>
<td>material is melted</td>
</tr>
<tr>
<td>transport</td>
<td>it is pushed into the mold</td>
</tr>
<tr>
<td>cooling</td>
<td>it is cooled</td>
</tr>
<tr>
<td>removal</td>
<td>the part is removed from the mold</td>
</tr>
</tbody>
</table>

There can be some overlap of the various steps.

Difference between and extruder and a plasticating unit for injection molding

extruder - a continuous melt stream leaves the unit

plasticating unit - has a reservoir to collect the melt until there is enough to make the parts (a shot)
For the best productivity
- parts are cooled as fast as possible
- residence time in flow channels should be minimized
  (to minimize material degradation)

One of the advantages of injection molding is that parts can be made to very
tight tolerances. This is not true of extrusion (in general).

To push the very viscous melt into molds where there might be small flow
channels and to get the tolerances required necessitates very high
pressures.

Pressures in an injection molding machine are on the order of 5000 to 10,000 psi
(as compared to 1000 – 2000 psi during extrusion).

Thus the machine has to be built to handle very large forces.
e.g. projected area of part is 10 in\(^2\)
then the clamping force required to hold the mold closed has to be higher
than the 100,000 lbf.

**Portions of Injection Molding Machine**

![Injection Molding Machine Diagram](image)
Sprue and runner are in the mold

multicavity part

There are 2 types of runner system
- cold runner (scrap)
- hot runner (melt has more exposure to high temperatures – also need to have some sort of shutoff in the gate area)
The clamping system can be either mechanical as is shown here.
or the clamping system can be hydraulic as is shown below.

If clamping is insufficient, the molds will be pushed apart and melt will leak out (flash)
Basic Injection Machines – 2 types
- reciprocating
- accumulator

Fig. The basic different injection plasticating (melting) system.
The following slide shows the basic cycle for reciprocating machines:

1. **Mold closes.** Screw begins moving forward for injection.
2. **Filling completes.** Screw continues moving forward for packing.
3. **Screw in full forward position.** Packing completes when gate freezes off.
4. **Cooling continues.** Screwback begins to accumulate melt ahead of screw tip for next shot.
5. **Cooling completes.** Mold opens for part ejection.
Mold Filling

The melt filling the mold is at a relatively high temperature and has a density less than at room temperature. As the part cools it begins to shrink. The material in the mold will no longer fill the entire cavity.

Fig. 2.5  The temperature dependence of the specific volume of polymers. (a) Amorphous. (b) Semicrystalline.
This can cause problems
- out of tolerance parts
- poor cooling
just to name two problems

Thus as the part is cooling, we need to keep adding more material. This is the **packing** phase of the cycle.

Once the gate area is closed or freezes off, we can add no more material. However the compressibility of the melt helps here.

Thus an injection molding cycle is as follows:

- fill
- pack
- cooling
- mold open
- part eject
- fill, mold open, etc.
- packing
- cooling

1 to 2 seconds
several seconds
several seconds to a few minutes

(Look at problem #1 on part shrinkage)
Part Cooling

- want to minimize cooling time (for economy)
- want to produce high quality parts
  - don’t want part deformation
  - don’t want crystallization
  - etc.

Some basic concerns for cooling
- How long will it take for the part to cool?
- Where do I put the cooling line?
- Is the coolant flow sufficient?

Cooling is done in the mold with the plastic in good contact (hopefully) with the metal of the mold.

Cooling models
- penetration model

\[
\frac{T - T_s}{T_0 - T_s} = \text{erf} \left( \frac{x}{2\sqrt{\alpha t}} \right)
\]

\[
t = \frac{x^2}{\alpha}
\]

\[
\frac{x_1}{2\sqrt{\alpha_1 t_1}} = \frac{x_2}{2\sqrt{\alpha_2 t_2}}
\]

\[
\frac{T - T_s}{T_0 - T_s} - \text{degree of cooling}
\]
cooling of a slab

Part Cooling - Slab

density
heat capacity
thermal conductivity
melt temperature
coolant temperature
Slab Cooling - Solution

OK because $\alpha_p << \alpha_s$
Another consideration for part cooling, is the decision on when to remove the part from the mold. In other words, what should be the part temperature when the part is OK to remove from the mold. One such choice is the freeze-off temperature. This is given for several polymers in the table below.

<table>
<thead>
<tr>
<th>Polymer</th>
<th>Apparent freeze-off temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>low-density polyethylene</td>
<td>90</td>
</tr>
<tr>
<td>polypropylene</td>
<td>135</td>
</tr>
<tr>
<td>general-purpose polystyrene</td>
<td>130</td>
</tr>
<tr>
<td>polycarbonate</td>
<td>200</td>
</tr>
<tr>
<td>nylon-6,6</td>
<td>240</td>
</tr>
<tr>
<td>PVC</td>
<td>140</td>
</tr>
<tr>
<td>PMMA (acrylic)</td>
<td>160</td>
</tr>
<tr>
<td>polyacetal</td>
<td>135</td>
</tr>
</tbody>
</table>
Some important parameters for cooling
- part thickness + variations
- thermal properties
- melt temperature
- removal temperature
- heat generated
- heat transfer to the mold

Cooling channel considerations
- flow should be turbulent for the best heat transfer
Part Cooling - Channel Analysis

Plate length 20 mm, all measurements in mm.
**Multicavity Molds**

We are continually looking for ways to increase productivity. An easy way to do this is by using multicavity molds. We can then make more than one part at a time.

**advantages**

- economy  
  (more parts per machine cycle)

**disadvantages**

- complicated melt delivery  
- complicated cooling system  
- quality can vary from cavity to cavity

Consider the following manifold system (melt delivery system)

Cavities 1 & 4 have longer flow paths than 2 & 3.

Cavities 2 & 3 will fill first. This could cause problems.

- property variations  
- hesitation of melt front and freezing of melt front leading to a partial fill
Need to balance the flow paths so that all parts fill at the same time.

Here (above) the lengths of the flow paths and diameters of successive runners (channels) is the same for each cavity.

Multicavity Molds
Balancing the Flow
Areas of Concern in Injection Molding

Part Filling

We need to fill the part fast enough so that the flow channels don’t freeze off before the part is filled (short shot). A faster fill means higher pressures plus possible other problems (shear heating, melt fracture, etc.).

Will the part fill? What kind of pressures are necessary?

How will the part fill? Is the manifold balanced?
- What are the flow patterns?
- Where are the weld lines?
- Is there orientation in the part?

Part Cooling

How long will it take?
Where do I put the cooling lines?
Is the coolant flow rate sufficient?

The cooling time is dependent upon the thickest section.

Use turbulent flow in the coolant channels in the mold.
Defects

Shrinkage Can Lead to Sink Marks

if the polymer skin is flexible when cold

semi-crystalline > amorphous shrinkage
as thickness ↑, shrinkage ↓

Shrinkage Can Lead to Voids

✓ if the polymer skin is stiff at room temperature

✓ induces weaknesses
✓ visible with transparent plastics
Shrinkage Can Lead to Residual Stresses

Warpage Can Occur Due to

- uneven shrinkage
- machining of a cold part after melt processing
  - stresses are no longer in equilibrium
  - the part deforms until equilibrium of the stresses is reestablished
Formation of Weld Lines

1. Melt fronts approach
2. Weld line forms
3. Meld line forms

Weld Lines with:
- a hole in the part
- multigate molds
Comparison of Extrusion Versus Injection Molding

<table>
<thead>
<tr>
<th>extrusion</th>
<th>injection molding</th>
</tr>
</thead>
<tbody>
<tr>
<td>continuous</td>
<td>cyclic</td>
</tr>
<tr>
<td>moderate pressures</td>
<td>high pressures (machines more costly)</td>
</tr>
<tr>
<td>dies are inexpensive</td>
<td>molds are very costly ($5M to $50M)</td>
</tr>
<tr>
<td>part dimensions may vary</td>
<td>tight tolerances on parts (e.g. finishes)</td>
</tr>
</tbody>
</table>

productivity depends upon
  - cooling time
    (keep the parts as thin as possible)
  - cavitation
    (make as many parts per cycle as possible)
  - operating window
    (need to operate in order to produce good quality parts)