Design with Plastics
Focus: Injection Molding

David O. Kazmer, P.E., Ph.D.
Department of Plastics Engineering
University of Massachusetts Lowell
One University Avenue
Lowell, Massachusetts 01854
david_kazmer@uml.edu
UMASS Lowell Plastics Engineering Department

- Nation’s only ABET accredited Plastics Engineering: B.S., M.S., E.D.
- 17 full-time faculty with decades of plastics experience.
- 4,000+ plastics engineering graduates placed in the plastics industry.
- Extensive plastics processing, testing, and design laboratory facilities.
Some useful reference information:


Kushmaul, Bill, What is a Mold, Techmold Inc., Tempe, AZ (1999)

*Standards and Practices for Plastics Molders* (Guidelines for Molders and Their Customers), Society of Plastics Industry, Washington DC.


Society of Plastics Engineers (good book list) 203-740-5475 or www.4spe.org

Hanser-Gardner (good book list) 800-950-8977 or www.hansergardner.com
Agenda

• Properties of Plastics
  – Nomenclature
  – Polymers: Structural vs. Molding
  – Morphology & Additives

• Process of Injection Molding

• Design for Injection Molding

• Case Study
Nomenclature

Plastic (adjective)
Plastics (noun)
Plastic Materials
Engineered Materials
Thermoplastics
Thermosets

All Plastics are Polymers

Polymer (poly + mer) = many + units
Plastics - “Polymers”

*Poly* (many) *Mer* (parts):
A large molecule made up of one or more repeating units (mers) linked together by covalent chemical bonds.

Example: polyethylene or poly(ethylene)

\[
\begin{align*}
n \text{ CH}_2 &= \text{ CH}_2 \\ &\xrightarrow{T, \ P} (\text{CH}_2 - \text{CH}_2)_n \\
\text{Monomer} &\quad (\text{ethylene gas}) \\ \text{Polymer} &\quad (\text{polyethylene}) \\
\text{n} &= \text{number of monomers reacting} >> 1
\end{align*}
\]
### Effect of Molecular Weight on the Properties of Polyethylene

<table>
<thead>
<tr>
<th>Number of -(CH₂ - CH₂ )- units (links)</th>
<th>Molecular weight (g/mol)</th>
<th>Softening temperature (°C)</th>
<th>Characteristic of the material at 23° C</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30</td>
<td>-169*</td>
<td>Gas</td>
</tr>
<tr>
<td>6</td>
<td>170</td>
<td>-12*</td>
<td>Liquid</td>
</tr>
<tr>
<td>35</td>
<td>1,000</td>
<td>37</td>
<td>Grease</td>
</tr>
<tr>
<td>140</td>
<td>4,000</td>
<td>93</td>
<td>Wax</td>
</tr>
<tr>
<td>250</td>
<td>7,000</td>
<td>98</td>
<td>Hard wax</td>
</tr>
<tr>
<td>430</td>
<td>12,000</td>
<td>104</td>
<td>Hard resin</td>
</tr>
<tr>
<td>750</td>
<td>21,000</td>
<td>110</td>
<td>Plastics</td>
</tr>
<tr>
<td>1,350</td>
<td>38,000</td>
<td>112</td>
<td>Hard resin</td>
</tr>
</tbody>
</table>

* * melting point
Must Balance Properties with Processability
Example: Polycarbonate
<table>
<thead>
<tr>
<th>Generallizations?</th>
<th>Amorphous vs. Semicrystalline Thermoplastics</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Amorphous (PC, PS, PVC…)</td>
</tr>
<tr>
<td></td>
<td>• Low mold shrinkage</td>
</tr>
<tr>
<td></td>
<td>• Limited chemical resistance</td>
</tr>
<tr>
<td></td>
<td>• Light transmission (many)</td>
</tr>
<tr>
<td></td>
<td>• High coefficient of friction</td>
</tr>
<tr>
<td></td>
<td>• Toughness or brittle?</td>
</tr>
<tr>
<td></td>
<td>• Stiff or flexible?</td>
</tr>
<tr>
<td></td>
<td>• Other properties ?</td>
</tr>
<tr>
<td></td>
<td>Semi-crystalline (PE, PP…)</td>
</tr>
<tr>
<td></td>
<td>• Higher mold shrinkage</td>
</tr>
<tr>
<td></td>
<td>• Good chemical resistance</td>
</tr>
<tr>
<td></td>
<td>• Opaque or translucent</td>
</tr>
<tr>
<td></td>
<td>• Low coefficient of friction</td>
</tr>
<tr>
<td></td>
<td>• Toughness (most)?</td>
</tr>
<tr>
<td></td>
<td>• Stiff or flexible?</td>
</tr>
<tr>
<td></td>
<td>• Other properties ?</td>
</tr>
</tbody>
</table>
## Common Additives for Plastics

<table>
<thead>
<tr>
<th>Colorants</th>
<th>Fillers</th>
</tr>
</thead>
<tbody>
<tr>
<td>UV Stabilizers</td>
<td>Reinforcements</td>
</tr>
<tr>
<td>Anti-oxidants</td>
<td>Anti-static Agents</td>
</tr>
<tr>
<td>Flame Retardants</td>
<td>Anti-microbial Agents</td>
</tr>
<tr>
<td>Internal Lubricants</td>
<td>Fragrances</td>
</tr>
<tr>
<td>External Lubricants</td>
<td>Plasticizers</td>
</tr>
<tr>
<td>Foaming Agents</td>
<td>Compatibilizing Agents</td>
</tr>
<tr>
<td>Other Plastics (blends)</td>
<td>etc........</td>
</tr>
</tbody>
</table>

Concentrations from PPM to 50% by weight
Steel:
- rigid
- strong
- tough

Glass Fiber Reinforced TP:
- rigid
- strong
- tough

Stress $F/A_o$

Strain $= \Delta L/Lo$

$E_{Steel} = 30,000,000$ psi

$E_{PC} = 1/100 \times E_{Steel}$

$E_{PC} = 300,000$ psi

(neat) Thermoplastic

Glass fibers (additive): stiffness $\uparrow$, strength $\uparrow$, toughness $\downarrow$, surface finish $\downarrow$, processability $\downarrow$, abrasive wear $\downarrow$, knit lines $\downarrow$, .......etc.
Agenda

• Properties of Plastics
• **Process of Injection Molding**
  – The Molding Cycle
  – Process Variants
• Design for Injection Molding
• Design for Assembly
• Case Study
Typical Modern Day Injection Molding Machine

**Clamp**
- open/close mold
- keep mold closed

**Mold**
- cavity+core
- with cooling

**Injection Unit**
- plasticate shot
- inject shot

**Hopper & Dryer**
“Low Pressure” Structural Foam Molding

For medium-large, thick parts

- low pressure (+)
- low warpage (+)
- few sinks (+)
- softer tool (+)
- surface splay (-)
- long cycle thick parts (-)
Multi-shot injection molding

Compatible materials: multi-color, hard / soft....
Incompatible materials: hinges, joints.....
Co-injection Molded Parts

- regrind / off-spec core
- barrier material core
- EMI / RF shielding
- reinforced core
- foamed core
- premium outer layer
- etc.
Gas Assist Injection Molding

Like co-injection molding, but second material is a “gas”.

“Contained Channel” GAIM:
Use to core out thick parts

“Open Channel” GAIM:
For conventional thickness parts
• Reduced warpage
• Lower fill pressures
“Metal” Injection Molding (MIM)

Metal Powder + Polymer Binder

→ Injection Mold Shape

→ Burn Off Binder and Sinter Metal
Packing and holding

Plastication and additional cooling

Part ejection
Agenda

• Properties of Plastics
• Process of Injection Molding
• Design for Injection Molding
  – Filling
  – Cooling
  – Ejection
• Design for Assembly
• Case Study
Injection Mold Filling

In practice, injection mold filing is non-isothermal

- Injecting “HOT” melt into “COLD” mold
- Injection times: 0.1 - 10 second range

Cooling of the melt at the cavity / core walls
Cold melt = high viscosity + high shear stress
Oriented material near the cavity walls solidifies

“Frozen-in” Orientation (2-Skins) + Random Core
Guidelines for Positioning Gates

1. Part Geometry
   “thick” to “thin”
   must allow venting
   equal pressure drop (balance)

2. Direction of Highest Stress in Use
   molecular orientation
   fiber orientation

3. Aesthetic Requirements
   gate vestige
   weld / knit lines

4. Dimensional Requirements
Gating “Scheme” - (Most) Important Decision

Gating Options: Many ! Best ?

Closed sleeve

Edge gate

Tunnel gate

Multiple edge gates

Top center gate

Multiple top gates
Gating from “thin to thick” will limit packing of the thicker section (sinks, voids......etc.)

*Should be avoided!*
Weld / Knit Lines

Gates

Knit line

Core

Single gate

Knit line
Start of mold filling

Weld / knit plane forms as flow fronts recombine
Weld line and failure due to flow around core.
Some Design Issues Related to Weld / Knit Lines

• Will the molded part have knit lines? If so,
• Where will the knit lines be located?
• Will the knit line areas have equivalent strength?
• Will the knit line areas be a cosmetic problem?
• Will the knit lines have equivalent chemical resistance?

*Filling simulations can provide “some” answers.*
## Typical Butt Weld Tensile Strength Retention Values (source LNP)

<table>
<thead>
<tr>
<th>Material Type</th>
<th>Reinforcement Type</th>
<th>Tensile Strength Retention (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polypropylene</td>
<td>no reinforcement</td>
<td>86%</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>20% glass fiber</td>
<td>47%</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>30% glass fiber</td>
<td>34%</td>
</tr>
<tr>
<td>SAN</td>
<td>no reinforcement</td>
<td>80%</td>
</tr>
<tr>
<td>SAN</td>
<td>30% glass fiber</td>
<td>40%</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>no reinforcement</td>
<td>99%</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>10% glass fiber</td>
<td>86%</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>30% glass fiber</td>
<td>62%</td>
</tr>
<tr>
<td>Polysulfone</td>
<td>no reinforcement</td>
<td>100%</td>
</tr>
<tr>
<td>Polysulfone</td>
<td>30% glass fiber</td>
<td>62%</td>
</tr>
<tr>
<td>PPS</td>
<td>no reinforcement</td>
<td>83%</td>
</tr>
<tr>
<td>PPS</td>
<td>10% glass fiber</td>
<td>38%</td>
</tr>
<tr>
<td>PPS</td>
<td>40% glass fiber</td>
<td>20%</td>
</tr>
<tr>
<td>Nylon 66</td>
<td>no reinforcement</td>
<td>91%</td>
</tr>
<tr>
<td>Nylon 66</td>
<td>10% reinforcement</td>
<td>89%</td>
</tr>
<tr>
<td>Nylon 66</td>
<td>30% reinforcement</td>
<td>60%</td>
</tr>
</tbody>
</table>
Guidelines for Weld Lines

1. Position welds in areas where the loads or stresses are “low” (via gating scheme).
2. Position welds in areas where visual or cosmetic demands are low (gating).
3. Disguise weld / knit line defect (texture…).
4. Keep melt temperature high (process).
5. Mold should be very well vented (tooling).
## Part Cooling

<table>
<thead>
<tr>
<th>Plate</th>
<th>Centerline reaches $T_e$</th>
<th>Average reaches $T_e$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$t_c$</td>
<td>$\frac{h^2}{\alpha \cdot \xi^2} \ln \left[ \frac{4}{\xi} \left( \frac{T_m - T_w}{T_e - T_w} \right) \right]$</td>
<td>$\frac{h^2}{\alpha \cdot \xi^2} \ln \left[ \frac{8}{\xi^2} \left( \frac{T_m - T_w}{T_e - T_w} \right) \right]$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cylinder</th>
<th>Centerline reaches $T_e$</th>
<th>Average reaches $T_e$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$t_c$</td>
<td>$0.173 \frac{R^2}{\alpha} \ln \left[ 1.6023 \left( \frac{T_m - T_w}{T_e - T_w} \right) \right]$</td>
<td>$0.173 \frac{R^2}{\alpha} \ln \left[ 0.6916 \left( \frac{T_m - T_w}{T_e - T_w} \right) \right]$</td>
</tr>
</tbody>
</table>

$t_c$ is the time required for the centerline temperature to reach the ejection temperature (s)

$t_a$ is the time required for the average part temperature to reach the ejection temperature (s)

$h$ is the wall thickness of a “plate-like” part (m)

$R$ is the radius of a “cylindrical” molding (m)

$T_m$ is the melt temperature at the start of cooling (°C)

$T_w$ is the cavity / core wall temperature during cooling (°C)

$T_e$ is the ejection temperature of the polymer (°C)

$\alpha$ is the thermal diffusivity of the polymer = $k / \rho c$ (m$^2$/s)

$k$ is the thermal conductivity of the polymer (W/m°C)

$c$ is the specific heat of the polymer (J/kg°C)

$\rho$ is the density of the polymer (kg/m$^3$)
Curve shape is material specific
poly-xxxxxxx

$t (2.0 \text{ mm}) = 3 \text{ to } 4 \ y (s)$

$t (1.0 \text{ mm}) = y (s)$

Typical melt temperature
Typical mold temperature

Part thickness (mm)

Part cooling time (seconds)
Molten amorphous polymer

Mold cooling

Molten semi-crystalline polymer (amorphous in the melt state)

Shrinkage due to thermal contraction only

Shrinkage due to thermal contraction and re-crystallization
"If" we could predict the cavity pressure - time curve to be used in molding (?) we could superimpose on the material’s P-v-T curve and predict volume shrinkage.

1-2 = filling
2-3 = packing
3-4 = p to h transfer
4-5 = hold
5 = gate freeze
6 = part size = cavity
7 = ejection
8 = ambient conditions
Dealing with “Area” Related Differential Shrinkage

- Lower mold shrinkage “near” the gate
- Using more gates leads to a more uniform mold shrinkage
- Mold cavity cut to compensate for differential mold shrinkage
“Warpage” due to differential “surface” shrinkage

Differential Cooling

Hot surface (insufficient cooling)

- Higher ejection temperatures
- Lower modulus materials
- Lower \( I \) value designs

Warpage (buckling)

Part ejection

Cool surface (adequate cooling)

- Lower ejection temperatures
- Higher modulus materials
- Higher \( I \) value designs

Internal stress (no buckling)

or
Thicker Sections = Hotter (more $\Delta T$) = More Shrinkage

Warpage due to the higher mold shrinkage of the thicker wall section

Utilize a more uniform wall thickness whenever possible
Core out thicker sections creating a more uniform part wall thickness and more uniform mold shrinkage

(a) \( L_2 < L_1 \) or \( L_3 \)

Area with greater wall thickness

(b) \( L_1 = L_2 = L_3 \)

Core out thicker sections creating a more uniform part wall thickness and more uniform mold shrinkage
“Sinks” form on surface opposite features such as ribs due to the increase local thickness and mold shrinkage.

- Uniform wall thickness at corner (best)
- Thick rib, proper radius
- Excessive radius / fillet
- Balanced rib and radius / fillet dimensions
- Potential areas for sink marks voids and shrinkage stress
- Thick corner section

Sink Marks
Some options when dealing with ribs, bosses ....

- (a.) “Recommended” proportions
- (b.) Disguise (texture)
- (c.) Core out “top”
- (d.) Core out “bottom”
- (e.) Foaming agent (struct foam)
- (f.) Gas assist molding
- (g.) Spread sink over more area?
Part Ejection

Injection - Packing - Holding - Cooling - Part Ejection

Design for Ejection is a very important aspect of Design for Manufacturability (DFM).

The plastic part design and tooling will be influenced by factors such as:

- the presence of undercuts
- fine features / details
- cavity / core draft angles
- surface finish requirements
- overall part size and complexity
- aesthetic requirements
Part ejection is a 2-step process:
(1) mold opens (2) ejector plate forward

“Camera View Finder” has a very complex geometry but was Designed for Ejection
Ejecting “Features”

Molded slots: no special mold actions required for part ejection

Molded sidewall hole: side action likely

Internal cantilever snap: no special mold action required when slot is used at base of beam

Internal cantilever snap: requires use of special mold action (lifter) to release the undercut hook
Rib Ejection: Adjacent E-pins, Blades, E-pin pads...
Cavity
Core

Sufficient sidewall draft required

Sidewall openings molded without any special mold action

Mold in open position

Shut off angle

Molded part
Core

Sufficient sidewall draft required
No special mold actions are required when snap beam is molded using the shut off method.
Ejecting Snap Fit Beams: Option 2 - *The Lifter*

Space for lifter movement during part ejection - no other design features can be located in the area.
Agenda

• Properties of Plastics
• Process of Injection Molding
• Design for Injection Molding
• Design for Assembly
  – Snap & Press Fits
  – Mechanical Fasteners
• Case Study
Design for Assembly (DFA)

- Minimize the number of parts required to produce a product by incorporating as many assembly features as possible into each part ($$$ savings).

  Fewer primary and secondary processes

- Avoid the use of “complicated” assembly techniques (snap >> self threading screw >> screw + insert >>…..).

- The saving in assembly cost must be balanced against the cost of more complicated tooling and primary molding operation.

Note: The quality of “assemblies” produced using competitive fastening methods / systems may not be equivalent. (e.g. snap fit assembly vs. self threading screw)
Snap Fits

Inseparable annular snap (90° return)

Separable annular snap joint
Lead-in angle

Snap Fits (Momentary Interference)

Deflection

Elastic recovery
Mechanical Fasteners (advantages)

- Operable (or reversible) joints or permanent assembly.
- An effective method for joining most thermoplastic & thermosetting parts (except very flexible items).
- Join parts produced in similar or dissimilar materials.
- Available in a variety of sizes and materials.
- The joining practices are very conventional.
- Metal “fastener’s” properties are independent of temp., time and RH (creep and ΔCTE can be a “joint” problem).
- The assembly strength is achieved quickly.
Mechanical Fasteners (limitations)

- Mechanical fasteners are point fasteners.
- Localized regions of potentially high stress.
- Holes >>> stress concentration and weld line formation.
- Thermal expansion mismatch.
- Additional pieces / parts.
- Gasket to achieve a fluid or gas tight seal.
**Machine screw and nut**
- Esthetic interruption on both top and bottom surfaces
- Many parts required for assembly
- Access to both top and bottom of part is required during assembly
- Need locking hardware to avoid vibration loosening
- Durable assembly

**Machine screw and insert**
- One clean smooth surface obtained
- Fewer parts required for assembly
- Internally threaded insert must be inserted into boss during or after molding
- Requires special equipment / tooling for insert
- Good overall durability
- Suitable for repeated assembly

**Self threading screw and plastic boss**
- One clean smooth surface obtained
- Minimum number of parts required for assembly
- Mating plastic threads formed during assembly
- Minimum fastener and equipment cost
- Limited durability (mating thread is plastic)
- Repeated assembly possible but limited
Type BT (25) thread cutting screw

Type B thread forming screw
Boss designs that result in the potential for sink marks and voids

Sinks / voids / cooling stresses

Improved Boss Designs

Boss attached to the wall using ribs

Thick sections cored out

Gussetts reinforce free standing bosses
Agenda

- Properties of Plastics
- Process of Injection Molding
- Design for Injection Molding
- Design for Assembly
- Case Study
  - Design review
  - Improved design
Case Study: PDA

• 500,000 units per year
• Injection molded top & bottom housing

• Rough concept design completed
• Improve design for performance & moldability
Top Design
Bottom Design
Case Redesign

- Rounded corners: large external & small internal
- Made same thickness (1.5 mm)
- Shifted parting plane to remove undercuts
- Added bevel to front
- Added ribbed boss & stand-off for mechanical assembly: Wall thickness at base 80% of nominal
- Improved spacing on bottom holes
- Identified gate & weld line locations, mold cavity layout
Top Redesign