**Injection moulding**

**Introduction**

Injection molding is generally used to produce thermoplastic polymers. It consists of heating of thermo plastic materials until it melts and then injecting into the steel mould, where it cools and solidifies to take its final shape. The plastic materials are usually received in the granular form. It is placed in the hopper of the moulding machine from which it is fed to a heated cylinder. Granules are heated in the cylinder to melt or plasticize. The typical melting point is about 180°C. The melting temperature varies with the material. The mould is usually made up of steel and it is water cooled. A plunger forces the molten plastics from the cylinder into the mould wherein, it cools and solidifies. The mould is opened and the moulded part as well as the attached runner is removed. Figure M2.4.1 shows the injection molding process.

![Injection Moulding Diagram](image-url)

**Figure M2.4.1: Injection molding**

**Typical characteristics of injection moulded parts**

Injection moulding is advantageous when it is required to produce the intricate parts in more quantity. Apart from the above, this method also can produce one moulded part that can replace an assembly of components. In injection moulding, parts can be often moulded directly with color and surface finish thereby avoiding secondary finish. Injection moulded parts are usually thin walled and heavy sections are usually not recommended. Since, thermoplastic parts are having less strength they are preferred to be used in lower stress application area. In the current practice, thermoplastics are reinforced with glass or other fibers and functionally competitive with Zinc and Aluminum.
Effect of shrinkage

Shrinkage is a common phenomenon all thermoplastic materials on cooling and solidification. Table M2.4.1 summarizes the shrinkage rate of common thermoplastic material upon solidification. Due to shrinkage the most common defect such as sink mark or surface depression occurs in the moulded part. The second common effect is the closing in on a U-shaped cross-section, particularly if reinforcing ribs are present. The third common type of effect is the occurrence of curvature on a flat surface in the direction of a boss, protuberance or added material. (Figure M2.4.2 to M2.4.4)

![Sink mark](image)

**Figure M2.4.2:** Typical sink mark opposite a heavy section.

![Closing in of U-shaped sections](image)

**Figure M2.4.3:** Shrinkage of plastic material on cooling

![Curvature in the flat surface](image)

**Figure M2.4.4:** Curving of flat surfaces caused by shrinkage of material.
Table M2.4.1: Shrinkage rate of common thermoplastic material upon solidification (Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)

<table>
<thead>
<tr>
<th>Thermoplastic</th>
<th>Percent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acetal</td>
<td>2.0–2.5</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.3–0.8</td>
</tr>
<tr>
<td>Acrylonitrile butadiene styrene</td>
<td>0.3–0.8</td>
</tr>
<tr>
<td>Nylon</td>
<td>0.3–1.5</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>0.5–0.7</td>
</tr>
<tr>
<td>Polyethylene</td>
<td>1.5–5.0</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>1.0–2.5</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>0.2–0.6</td>
</tr>
<tr>
<td>Polyvinyl chloride, rigid</td>
<td>0.1–0.5</td>
</tr>
<tr>
<td>Polyvinyl chloride, flexible</td>
<td>1.0–5.0</td>
</tr>
</tbody>
</table>

**Suitable materials**

Commonly used thermoplastic materials are

- Polyethylene
- Polypropylene
- Polystyrene
- Polyvinyl chloride (vinyl or PVC)
- Nylon
- Acrylonitrile butadiene styrene (ABS)
- Acrylic
Table M2.4.2: Commonly used thermoplastic materials used for injection moulding (*Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed*).

<table>
<thead>
<tr>
<th>Material</th>
<th>Tensile strength (MPa)</th>
<th>Maximum Service Temperature (°C)</th>
<th>Specific Gravity</th>
<th>Application</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polystyrene</td>
<td>48</td>
<td>82</td>
<td>1.04</td>
<td>Toys, containers</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>34</td>
<td>110</td>
<td>0.91</td>
<td>House wares</td>
</tr>
<tr>
<td>High density Polyethylene</td>
<td>28</td>
<td>125</td>
<td>0.96</td>
<td>Refrigerator parts, House wares</td>
</tr>
<tr>
<td>PVC</td>
<td>21</td>
<td>82</td>
<td>1.4</td>
<td>Seals, Electrical plugs</td>
</tr>
</tbody>
</table>

**Design recommendations**

These are the following design recommendations used for injection moulded parts:

*Gate and ejector pin location:* Surface finish of the part depends upon the location of gate and ejector pin. Ejector pins are usually located on the underside of the part if it has both underside and outside. Gates can be located in a number of locations as shown in Figure M2.4.5.
In case of round and cylindrical parts center gating is preferred and for large area parts near center-gating is recommended.

**Figure M2.4.5:** Various gating systems

**Figure M2.4.6:** Maintain uniform wall thickness and avoid abrupt change in wall thickness and make it gradual change
### Table M2.4.3: Various wall thicknesses for different thermoplastic materials with changing cross-section as shown in Figure M2.4.6. (Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)

<table>
<thead>
<tr>
<th>Material</th>
<th>Short sections</th>
<th>Small sections</th>
<th>Average sections</th>
<th>Large sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acetal</td>
<td>0.6</td>
<td>0.9</td>
<td>1.9</td>
<td>3.2–4.7</td>
</tr>
<tr>
<td>Acrylic</td>
<td>0.6</td>
<td>0.9</td>
<td>2.3</td>
<td>3.2–6.3</td>
</tr>
<tr>
<td>Acrylonitrile butadiene styrene</td>
<td>0.9</td>
<td>1.3</td>
<td>1.9</td>
<td>3.2–4.7</td>
</tr>
<tr>
<td>Cellulose acetate butyrate</td>
<td>0.6</td>
<td>1.3</td>
<td>1.9</td>
<td>3.2–4.7</td>
</tr>
<tr>
<td>Nylon</td>
<td>0.3</td>
<td>0.6</td>
<td>1.5</td>
<td>2.4–3.2</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>0.4</td>
<td>0.8</td>
<td>1.8</td>
<td>2.4–3.2</td>
</tr>
<tr>
<td>Polyethylene</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Low-density</td>
<td>0.9</td>
<td>1.3</td>
<td>1.6</td>
<td>2.4–3.2</td>
</tr>
<tr>
<td>High-density</td>
<td>0.9</td>
<td>1.3</td>
<td>1.9</td>
<td>3.2–4.7</td>
</tr>
<tr>
<td>Polypropylene</td>
<td>0.6</td>
<td>0.9</td>
<td>1.9</td>
<td>3.2–4.7</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>0.8</td>
<td>1.3</td>
<td>1.6</td>
<td>3.2–6.3</td>
</tr>
<tr>
<td>Polyvinyl chloride</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flexible</td>
<td>0.6</td>
<td>1.3</td>
<td>1.9</td>
<td>3.2–4.7</td>
</tr>
<tr>
<td>Rigid</td>
<td>0.9</td>
<td>1.6</td>
<td>2.4</td>
<td>3.2–4.7</td>
</tr>
</tbody>
</table>

**Suggested wall thickness:** Walls should be of uniform thickness as possible. When changes in the wall thickness are unavoidable, the transition must be gradual instead of abruptly. Thinner walls are more feasible with small parts.

**Holes:**

- Holes are feasible in injection moulded parts, but knit or weld lines are often developed adjacent to the holes. Flashing may occur at the edge of the hole.

- The minimum spacing between the two adjacent holes or the spacing between the edge and the hole should be one diameter(d) of the hole.(Figure M2.4.7)
**Figure M2.4.7**: Minimum spacing for holes and sidewalls.

- Minimum distance between a hole and edge of the part should be three diameters (d) or more. (See Figure M2.4.8)

**Figure M2.4.8**: Minimum spacing for holes and sidewalls.

- Through hole is preferred than a blind hole
- Hole in the bottom of the part is preferable than the side
- Blind hole should not be more than two diameters deep. If the diameter is 1.5 mm or less, then one diameter is recommended. (see Figure M2.4.9)

**Figure M2.4.9**: Recommended depth limits for blind holes.

- To increase the depth of a blind hole steps are used thereby enabling employment of stronger core pin. (Figure M2.4.10)
For through holes, cutout sections in the part can shorten the length of a small-diameter pin. (Figure M2.4.11)

Overlapping and offset mould-cavity projections are preferred instead of core pins for producing holes parallel to the die-parting line (perpendicular to the mould-movement direction). This approach is explained through Figures M2.4.12 and M2.4.13 which is applicable to injection moulding as well as to die castings.

Figure M2.4.10: If a blind hole must be deep, use a stepped diameter.

Figure M2.4.11: Improved design on the right provides better rigidity of the mould core pin.

Figure M2.4.12: A die casting’s wall slopes sufficiently, through holes which is formed by using “kissing cores” built into opposite die halves.
**Ribs:**

- Reinforcing ribs should be thinner than the wall they are reinforcing to avoid sink marks. It is suggested that the rib thickness should be between 40% and 60% of wall thickness.

- In order to provide additional reinforcement, two ribs may be used if necessary and the rib should be two or more wall thickness apart.

- Ribs need to be provided perpendicular to the parting line

- A generous draft of 0.5° to 1.5° per side is recommended for ribs.

- There should be a radius at the base of 25% to 40% of the wall thickness as shown in Figure M2.4.14.

**Bosses:**

Bosses are protruding pads designed to provide mounting surfaces or reinforcements around the hole. The above mentioned recommendations for maximum height, draft and radius of
ribs are generally applicable to solid or hollow bosses. Bosses in the upper portion of the mould are avoided because these might trap gases. It is preferred to locate bosses in the corners to help the material flow in filling the mould.

**Undercuts:**

In this process undercuts are possible but it requires sliding cores or split moulds. In fact, external undercuts can be used at the parting line or extended to the line to obviate the need for core pull. The requirement of core pull can be avoided by providing shallow undercuts which often may be strippable from the mould. Furthermore, if such undercuts are strippable, the other half of the mould needs to be removed first. Then the mould ejector pins can act to strip the part. Figure M2.4.15 shows the average maximum strippable undercut for common thermoplastics.

![Diagram of undercuts](image)

Figure M2.4.15: Allowable undercut for common materials.

**Screw threads:**

- To mould screw thread, a core is used that can be rotated after the completion of the moulding cycle. This basically unscrews the part and helps it to remove from mould.

- Axis of the screw are put on the parting line thereby avoiding the need for a rotating core. This method is only applicable for external threads. (see Figure M2.4.16)

- The threads are made few, shallow and of rounded forms so that the part can be stripped from the mould without unscrewing as shown in Figure M2.4.17.
**Figure M2.4.16:** Axis of the screw is placed on the parting line to avoid the need of rotating the core. *(Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)*

**Figure M2.4.17:** Threads are made shallow in order to strip the part easily from the mould. *(Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)*

**Inserts:**

Sharp corners need to be avoided in inserts which are immersed in the thermoplastics. Knurls on the machined inserts should be relatively coarse to permit the material to flow into the recess. A smooth surface has to be provided where the insert exits from the plastic. Boss generally provides supporting material for inserts and hence often inserts are incorporated in the boss. To do so, if the outside diameter of the insert is less than 6 mm, outside diameter of the boss should be twice that of the insert otherwise i.e. if the outside diameter of the insert is larger than 6 mm, then wall thickness should be 50 to 100 percent of the insert diameter. As a thumb rule the embedded length of an insert is considered to be twice its diameter. (See Figure M2.4.18 and Figure M2.4.19)

**Figure M2.4.18:** The depth of insertion should be at least twice that of insert diameter.
Lettering and surface decoration:

Lettering is incorporated in the mould. Lettering in a part should be raised with machined mould. With hubbed dies it is best to have depressed letters. But in both the cases i.e. raised as well as depressed, the letters should be perpendicular to the parting line. Cavities for filling lettering should be sharp edged and 0.13 to 0.8mm wide as shown in Figure M2.4.20. They should have one half as deep as wide.

Draft:

It is strongly recommended to provide a minimum draft in the side wall of the injection moulded part to allow easy removal of the moulded parts from the mould. The recommended drafts for some commonly used materials as shown in the Table M2.4.4.
Table M2.4.4: List of Material and corresponding draft allowance (Source: Design for Manufacturability Handbook by James G Bralla, 2nd Ed)

<table>
<thead>
<tr>
<th>Material</th>
<th>Draft</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyethylene</td>
<td>1/4°</td>
</tr>
<tr>
<td>Polystyrene</td>
<td>1/2°</td>
</tr>
<tr>
<td>Nylon</td>
<td>0–1/8°</td>
</tr>
<tr>
<td>Acetal</td>
<td>0–1/4°</td>
</tr>
<tr>
<td>Acrylic</td>
<td>1/4°</td>
</tr>
</tbody>
</table>

**Corners:**

Sharp corners are recommended to be avoided except at the parting line. Major problem with sharp corners is that, these obstruct the smooth flow of material and create surface defect. Also, sharp corners are the point of stress concentration. Hence, it is strongly recommended to provide radii as generous as possible. A minimum of 0.5mm radius has to be provided under any situation and 1 mm is preferable if the part permits.

**Surface finish:**

It is possible to achieve a high glossy surface finish, if the mould is highly polished. Even though Painting of most thermoplastics is feasible but is not recommended if the colour can be moulded into the parts. The latter approach is more economical and gives better results. Parting line flash can be easily removed by giving a gap between the surface decoration (reeds, textures & flutes) and the parting line (Figure M2.4.21)

![Figure M2.4.21: Easy removal of the parting line flash by giving a gap to surface decoration from parting line.](image)
**Flat surface:**

The gently curved surface is preferred than flat surface as it is more prone to the formation of irregularities. Curved surface produces more rigid parts.

**Mould parting line:**

The part and the mould should be designed in such a way that the parting line occurs in such an area so that it doesn’t adversely affect the appearance or function of the part. It is preferred to keep the parting line at the edge of the part where already a sharp edge exists. But it is important to note that removal of parting-line flashing might destroy the sharpness of the corner. If it is not possible to keep the parting line at the edge, a bead on the parting line facilitates removal of the mould flash. Parting line should be straight. (Figure M2.4.22 and Figure M2.4.23)

![Mould parting line diagram](image)

**Figure M2.4.22:** A bead on the parting line facilitates removal of mold flash.

![Mould parting line diagram](image)

**Figure M2.4.23:** Deliberately offset sidewalls help avoid appearance defects if mould halves do not line up properly.

**Dimensional factors and tolerance**

Dimensions can’t be achieved with close tolerance because of the following reasons

- Material shrinkage
- Variation of the process parameter from cycle to cycle
- Position of the runner, cooling channel and gate
Different plastics materials have different tolerance capabilities. Materials having a low shrinkage allowance can be molded with close tolerance.

The allowable dimensional tolerances increases by 5 percent for each cavity as the number of mould cavities are more according to the thumb rule. For example, a single cavity mould with an allowable tolerance of ±0.1 mm on a particular dimension should have ±0.15 mm if the number of cavities is 10 (10×5 percent = 50 percent increase in tolerance).