2 – Injection Moulding

The Process and Equipment

Because most engineering thermoplastic parts are fabricated by injection moulding, it is important for the designer to understand the moulding process, its capabilities and its limitations.

The basic process is very simple. Thermoplastic resins such as DELRIN® acetal resins, CRASTIN® and RYNITE® thermoplastic polyester resins, or ZYTEL® nylon resins, supplied in pellet form, are dried when necessary, melted, injected into a mould under pressure and allowed to cool. The mould is then opened, the parts removed, the mould closed and the cycle is repeated.

Fig. 2.01 is a schematic of the injection moulding machine.

Fig. 2.02 is a schematic cross section of the plastifying cylinder and mould.

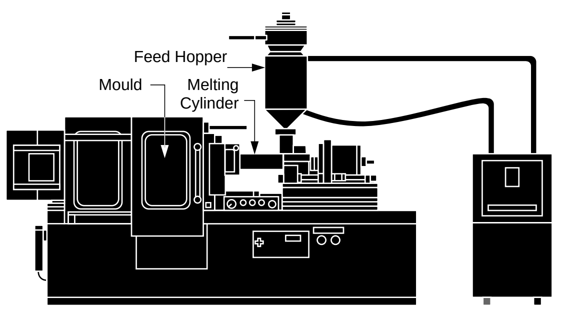

Fig. 2.01 **Injection moulding machine**

The Moulding Machine

Melting the plastic and injecting it into the mould are the functions of the plastifying and injection system. The rate of injection and the pressure achieved in the mould are controlled by the machine hydraulic system. Injection pressures range from 35-140 MPa. Melt temperatures used vary from a low of about 215°C for DELRIN® acetal resins to a high of about 300°C for some of the glass reinforced ZYTEL® nylon and RYNITE® polyester resins.

Processing conditions, techniques and materials of construction for moulding DuPont Engineering Thermoplastic Resins can be found in the Moulding Guides available for DELRIN® acetal resins, MINLON® engineering thermoplastic resins, CRASTIN® and RYNITE® thermoplastic polyester resins and ZYTEL® nylon resins.

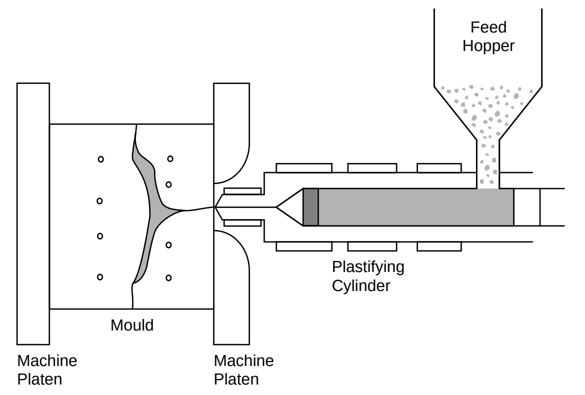

Fig. 2.02 **Plastifying cylinder and mould**

The Mould

Mould design is critical to the quality and economics of the injection moulded part. Part appearance, strength, toughness, size, shape, and cost are all dependent on the quality of the mould. Key considerations for Engineering Thermoplastics are:

- Proper design for strength to withstand the high pressure involved.
- Correct materials of construction, especially when reinforced resins are used.
- Properly designed flow paths to convey the resin to the correct location in the part.
- Proper venting of air ahead of the resin entering the mould.
- Carefully designed heat transfer to control the cooling and solidification of the mouldings.
- Easy and uniform ejection of the moulded parts.

When designing the part, consideration should be given to the effect of gate location and thickness variations upon flow, shrinkage, warpage, cooling, venting, etc. as discussed in subsequent sections. Your DuPont representative will be glad to assist with processing information or mould design suggestions.

The overall moulding cycle can be as short as two seconds or as long as several minutes, with one part to several dozen ejected each time the mould opens. The cycle time can be limited by the heat transfer capabilities of the mould, except when machine dry cycle or plastifying capabilities are limiting.

Trouble Shooting

In case moulded parts do not meet specifications, the reasons need to be detected. Table 2 shows a list of basic solutions to general moulding problems.

For more details contact DuPont's Technical Service.

Trouble Shooting Guide for Moulding Problems

Problem	Suggested Corrective Action(s)
Short shots, poor surface finish	1. Increase feed. 2. Increase injection pressure. 3. Use maximum ram speed. 4. Decrease cushion. 5. Raise material temperature by raising barrel temperature. 6. Raise mould temperature. 7. Increase overall cycle. 8. Check shot size vs. rated machine shot capacity; if shot size exceeds 75% of rated (styrene) shot capacity, move to larger machine. 9. Increase size of sprue and/or runners and/or gates.
Flashing	1. Lower material temperature by lowering barrel temperature. 2. Decrease injection pressure. 3. Decrease overall cycle. 4. Decrease plunger forward time. 5. Check mould closure (possible obstruction on parting line surface). 6. Improve mould venting. 7. Check press platens for parallelism. 8. Move mould to larger (clamp) press.
Nozzle drool	1. Lower nozzle temperature. 2. Lower material temperature by lowering barrel temperature. 3. Decrease residual pressure in barrel by: a) reducing plunger forward time and/or back pressure; b) increasing 'decompress' time (if press has this control). 4. Decrease die open time. 5. Use nozzle with positive shut-off valve.
Nozzle freeze-off	1. Raise nozzle temperature. 2. Decrease cycle time. 3. Increase injection pressure. 4. Raise mould temperature. 5. Use nozzle with larger orifice.
Discolouration	1. Purge heating cylinder. 2. Lower material temperature by lowering barrel temperature. 3. Lower nozzle temperature. 4. Shorten overall cycle. 5. Check hopper and feed zone for contaminants. 6. Check barrel and plunger or screw fit for excessive clearance. 7. Provide additional vents in mould. 8. Move mould to smaller shot size press.

Trouble Shooting Guide for Moulding Problems (continued)

Problem	Suggested Corrective Action(s)
Burn marks	1. Decrease plunger speed. 2. Decrease injection pressure. 3. Improve venting in mould cavity. 4. Change gate location to alter flow pattern.
Brittleness	1. Pre-dry material. 2. Lower melt temperature and/or residence time. 3. Raise mould temperature. 4. Reduce amount of regrind.
Sticking in cavities	1. Decrease injection pressure. 2. Decrease plunger forward time, packing time/pressure. 3. Increase mould closed time. 4. Lower mould temperature. 5. Decrease barrel and nozzle temperature. 6. Check mould for undercuts and/or insufficient draft. 7. Use external lubricants.
Sticking in sprue bushing	1. Decrease injection pressure. 2. Decrease plunger forward time, packing time/pressure. 3. Increase mould closed time. 4. Increase mould temperature at sprue bushing. 5. Raise nozzle temperature. 6. Check sizes and alignments of holes in nozzle and sprue bushing (hole in sprue bushing must be larger). 7. Provide more effective sprue puller.
Weld lines	1. Increase injection pressure. 2. Increase packing time/pressure. 3. Raise mould temperature. 4. Raise material temperature. 5. Vent the cavity in the weld area. 6. Provide an overflow well adjacent to the weld area. 7. Change gate location to alter flow pattern.
Sinks and/or voids	1. Increase injection pressure. 2. Increase packing time/pressure. 3. Use maximum ram speed. 4. Raise mould temperature (voids). 5. Lower mould temperature (sinks). 6. Decrease cushion. 7. Increase size of sprue and/or runners and/or gates. 8. Relocate gates nearer thick sections.
Warpage/ part distortion	1. Raise tool temperature, uniform? 2. Increase gate and runner size. 3. Increase fill speed. 4. Increase injection pressure and packing time/pressure. 5. Check flow path and relocate gate position and/or amend part design.

Trouble Shooting Guide for Moulding Problems (continued)

Problem	Suggested Corrective Action(s)
Poor dimensional control	1. Set uniform cycle times. 2. Maintain uniform feed and cushion from cycle to cycle. 3. Fill mould as rapidly as possible. 4. Check machine hydraulic and electrical systems for erratic performance. 5. Increase gate size. 6. Balance cavities for uniform flow. 7. Reduce number of cavities.