

INTRODUCTION TO THERMOFORMING - Injection Moulding

5

INTRODUCTION

From the moment we get up in the morning until the moment we go to bed at night we are surrounded by products that have been produced, wholly or partially, on Injection Moulding Machines. The alarm clock, shower head, hair brush, coffee machine, toaster, toothbrush – even the buttons on your blouse or shirt, owe their existence in their current form to Injection Moulding. Outside of our homes injection moulded products are still all around us – the car, bus or train you ride to work, school or college is full of injection moulded components and whatever you do, there's a good chance that you will spend a large portion of your day tapping the injection moulded keys of an injection moulded computer or holding the injection moulded hand piece of an injection moulded phone or writing things down with an injection moulded biro. At the end of the day, many of us watch television screens that are encased in injection moulded plastic; often changing the channels with the injection moulded remote control that we hold in our hands. Even when we go to bed at the end of the day, if we look at the switch we use to turn out the light, whether it is on the wall, or in the lamp on the bedside table, or screwed to the head board; it is a piece of injection moulded plastic.

Injection Moulding is an important part of our every day lives, our world would be very different without it and product designers need to know about it; they will use it many times during their careers.

Injection Moulding is the process of heating plastic granules to melting point before injecting them at high pressure through a nozzle into a mould. When the plastic cools the mould is opened and the newly formed plastic part is removed.

The process has been modified and developed in numerous ways and now there are many different types of Injection Moulding, such as:

- Injection Blow Moulding
- Twin/Triple Injection Moulding
- Multi-component injection moulding
- Multi-station injection moulding
- Reaction injection moulding
- Gas injection moulding – and many more.

Both Thermoplastics (including Thermoplastic Elastomers, the Thermoplastic 'rubber') and Thermosetting plastics are injection moulded to produce an enormous and ever increasing range of products and components.

EVOLUTION OF INJECTION MOULDING

One of the earliest forms of plastic moulding was Compression Moulding. Here, a fixed amount of plastic is placed in the lower half of a mould and heated before the upper half of the mould is closed over the top of it. The mould remains closed while the part cools and when it is taken off the 'flash' (excess material that seeps between the two halves of the mould) is removed.

Transfer Moulding introduces a plunger, or ram, that pushes the plastic through a barrel and into the mould cavity, which is already closed. Transfer Moulding reduces the amount of waste and removes the need for de-flashing. Some waste material is still produced though, in the barrel and interconnecting parts of the mould (depending on its shape).

Plunger Moulding has the plunger mounted horizontally and the plastic fed into the barrel from a hopper mounted on top. As the plunger moves along the barrel it automatically cuts off the supply of granules, leaving a fixed amount of material in the barrel for injecting. The barrel has a nozzle at its end that connects to the mould and the mould itself has a 'sprue' or narrow channel through which the plastic moves on its way to the mould cavity.

The last major innovation, which led to the Injection Moulding Machine, was the extruding, Archimedean or plasticising screw. The Screw is a tapered bar with a spiral of flights along its length and was first used, not surprisingly, for extruding. The height and pitch of the flights varies according to which of the three zones of the screw they occupy and hence, which job they are designed to do as the screw turns: Feeding, Compressing or Metering.

1. The Feed zone flights are long and designed to move the material along the barrel as quickly as possible. As they move along, the granules are heated by friction (from the movement of the screw inside the barrel and from the movement of the granules themselves) and by the inside of the barrel against which the granules are forced by movement of the flight.
2. In the Compression zone the flights become shorter and the plastic granules are further heated and compressed, removing any air pockets. The plastic is melted by now and becomes thoroughly mixed or homogenized by the continuing movement of the screw.
3. The Metering zone contains the shortest flights, which are designed to pump the plastic or melt, through the extrusion die.

Various methods of incorporating Archimedean screw extruders into injection moulding machines were developed until the most efficient was found; the Reciprocating Screw. Basically the screw is capable of moving back and fore as well as turning so that it can act as a plunger and an extruder. As melt builds up at the end of the screw, the screw moves backwards. Once there is enough melt to fill the mould (measured by how far back the screw has traveled) the screw is driven forward, pushing the melt through the nozzle, along the sprue and into the mould cavity. A non-return valve is fitted at the end of the screw to prevent the melt from being pushed back along the screw.

The evolution of injection moulding was driven by ever-greater demand for plastic products as their use became more widespread. Reciprocating Screw Injection moulding machines had three important advantages over their predecessors, the plunger moulding machines:

1. Output. The injection moulding machine dramatically increased the amount of melt, and thus the amount of product, that could be

produced.

2. Quality. The melt produced by the screw is more homogenized and the heating more thorough (unmelted particles, common in plunger moulding, are never encountered in injection moulding) thus improving the quality and consistency of products.
3. Energy efficiency. The heat generated by the friction of all the moving components inside the barrel greatly reduces the amount of external heating required. In fact, some thermoset injection moulding machines are fitted with cooling mechanisms because too much friction heat is generated.

The development of Injection Moulding machines at the beginning of the 21st Century sees a move towards all electric machines. Wherever possible, expensive, high maintenance hydraulics are replaced by servo-motors which reduce the size and weight of machines and improve accuracy, speed and consistency.

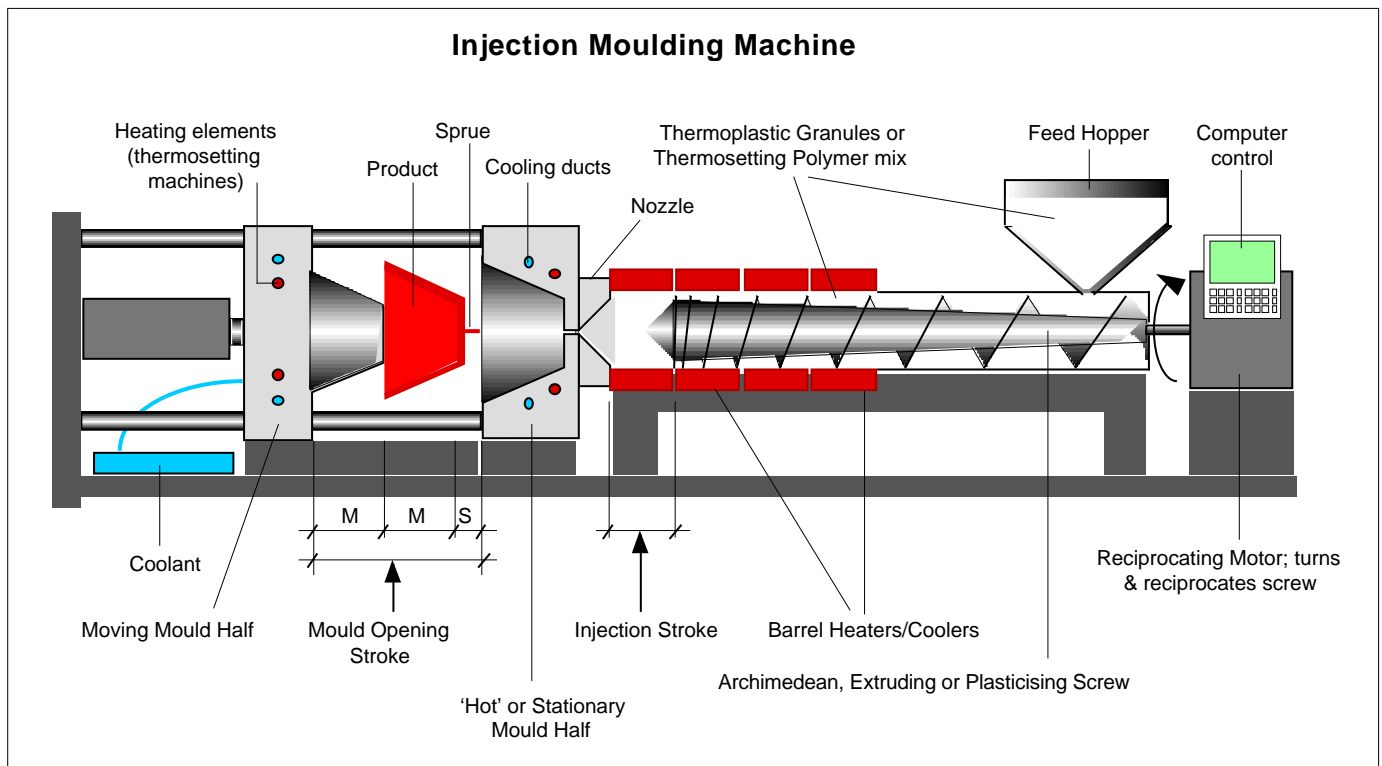
INJECTION MOULDING MACHINE BASICS

Injection Moulding Machines have become highly sophisticated, complex, computerized machines with many features that need to be considered when deciding which machine will be able to make a particular product. Here, we will talk about some of the main features, how they effect output and, in some cases, the rules that are used to determine these effects.

SCREW

The characteristics of the screw are crucial in determining the type and size of product that can be produced on an Injection Moulding Machine and many machines are supplied with more than one. One of the most important factors is the ratio between the length and the diameter, or the L/D ratio.

22:1 or higher (meaning that the screw is 22 times the length of the diameter) improves the mix of the melt and the quality of the product. Compared to a screw of the same length with a lower L/D ratio of perhaps 20:1, the injection pressure will be higher but the volume lower. The volume can be increased with a longer injection stroke but this increases the cycle time.



Generally speaking for high quality products such as engineering components, a L/D ratio of 22:1 or higher is needed. 20:1 is suitable for medium quality products like garden furniture and 18:1 would typically be used for low quality items such as disposable packaging or cheap children's toys.

INJECTION PRESSURE

This is the pressure in the barrel at the point of injection, expressed in kg/cm² or kbar. A higher L/D ratio produces greater Injection Pressure. The greater the Injection Pressure the better the quality of product produced.

SHOT WEIGHT

The Shot Weight of a machine is defined as the weight of plastic produced at the nozzle in a normal cycle (without a mould) in PS (specific gravity 1.05). The shot weight for other materials can be worked out by the following formula:

$$X \times \frac{Y}{1.05} \quad \begin{cases} X = \text{Specific Gravity of materia.} \\ Y = \text{Shot Weight of machine} \end{cases}$$

The shot weight of a machine does not tell us the maximum volume of part that that machine can produce, because the pressures for moulding are higher than for extruding. Typically, the maximum volume of a low quality part will be 85% of the Shot Weight, or around 75% for a high quality part.

Parts need to weigh between 35% and 85% of the Shot Weight of a machine, any lower and the machine may be damaged during production, any higher and the mould cavity or cavities will not be filled.

INJECTION STROKE

The distance that the screw travels during an injection stroke, up to 4 * diameter.

INJECTION VOLUME

Theoretically, the injection volume is the length of the injection stroke multiplied by the cross sectional area of the screw. However, leakage around the screw and movement of the non-return valve mean that in reality, only around 90% of that Injection Volume gets injected. If Injection Volume is used to determine the size of machine needed for a specific part (as opposed to using Shot Weight) then for low quality parts the volume should be within 20 – 80% of the Injection Volume, or within 40 – 60% for high quality parts.

INJECTION SPEED

Two things need to be considered when deciding how fast a product should be injected. Firstly, there needs to be enough speed for the mould to fill before the polymer cures or the melt un-melts. Thin walled parts in particular need fast injection speeds; the narrower a cavity is, the less time is available to fill it.

The other factor is the Constant Melt Front theory, which holds that optimum product quality is attained when the leading edge of the melt (or the Melt Front) travels at a constant speed through the mould cavity. Most parts have varying cross-sections that require different injection speeds in order to move the Melt Front at a constant speed. So whilst a machine will have a maximum injection speed, it will also have a number of other speeds below this, which may be used in any given cycle to try and achieve a constant Melt Front speed.

INJECTION RATE

This is the volume of melt produced by the screw, expressed in cm³ per second.

SCREW ROTARY SPEED

The Screw Rotary Speed (SRS) determines the Screw Surface Speed (SSS), which is another important factor. Polymer manufacturers specify maximum Screw Surface Speeds for each material that they produce and so, using the screw diameter, the maximum Screw Rotary Speed must be determined in order not to exceed this.

For example, imagine we were going to Injection Mould a mobile phone cover from PP. The maximum SSS of PP is 850 – but the optimum is 750. (Because we want the phone cover to be the best quality possible we will use this figure, rather than the maximum).

$$750 = \frac{\pi}{4} \times 50 \times \frac{SRS}{60}$$

$$\frac{750 \times 60}{\pi \times 50} = SRS = 286 \text{ rpm}$$

The screw diameter is 50 mm so the Screw Rotary Speed is:

CLAMP FORCE

The melt is injected into the mould under very great pressure, which the mould halves, and any other components or cores, must be able to resist in order to maintain their shape and to avoid any seepage. The larger a component is, the greater the pressure and so, the greater the Clamp Force required. The Clamp Force is often used in the machine name, for example an Injecto 50 would be an Injection Moulding machine with a 50 ton clamp force capability.

The clamp force required for a given part can be calculated once the shape and size of the part is known and some other factors are taken into account.

To calculate the clamp force you can use – the projected area (area of a moulded part projected onto a plane at right angles to the direction of the mould) multiplied by a constant, as denoted (for example) in the following table:

Thermoplastic	Tons/in ²	Tons/cm ²	MN/m ²
HIPS	1.0 – 2.0	0.155 – 0.31	15.4 – 30.9
HIPS (thin walled)	2.5 – 3.5	0.388 – 0.543	38.6 – 54.0
ABS	2.5 – 3.0	0.388 – 0.62	38.6 – 61.8
HDPE	1.5 – 2.5	0.233 – 0.388	23.2 – 38.6
PP	1.5 – 2.5	0.233 – 0.388	23.3 – 38.6
Acrylic	2.0 – 4.0	0.31 – 0.62	30.9 – 61.8
PC	3.0 – 5.0	0.465 – 0.775	46.3 – 77.2

So the clamping force for a thin walled PS product with a Projected Area of 120 cm² would be:

$$120 \times 0.388 = 46.56 \text{ tons minimum or } 120 \times 0.543 = 65.2 \text{ tons maximum.}$$

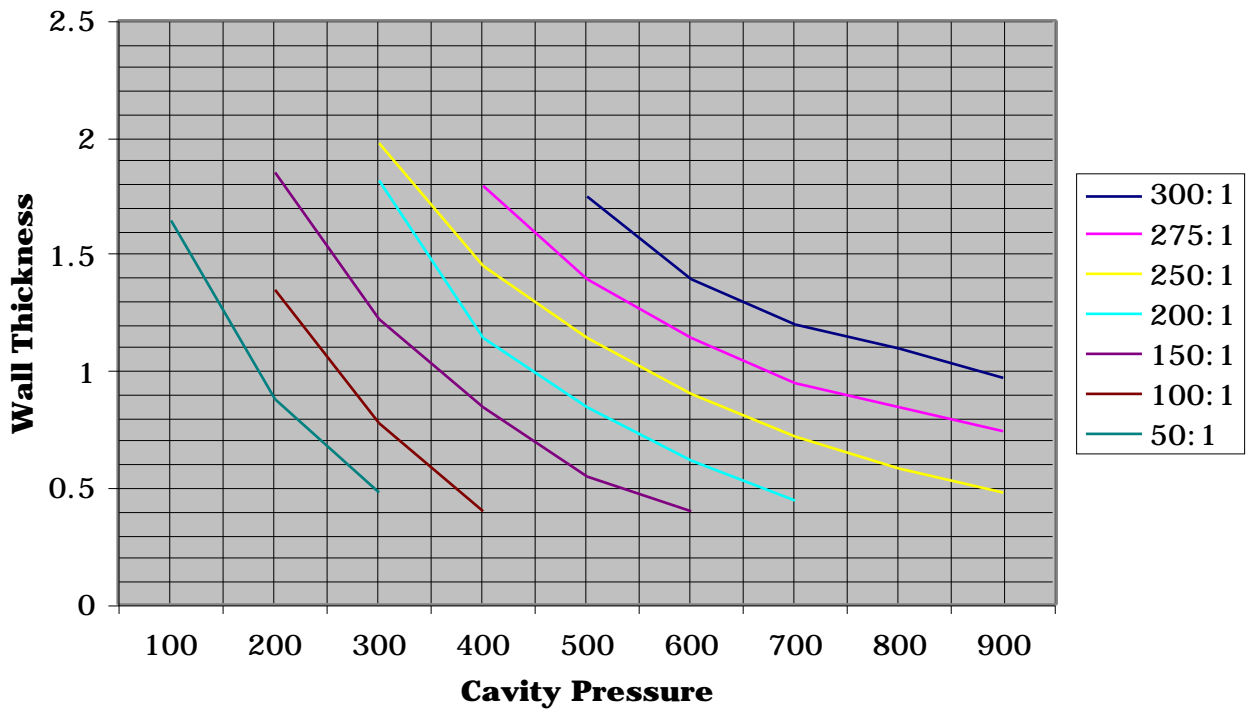
Perhaps a 50 ton machine will do, or we might need a 60 ton machine.

To help us decide, we can use the Flow Path Length (the distance from the centre of the end of the nozzle to the part of the mould furthest from that point) along with a couple of useful tables.

Unit Conversions

1 N	1/9.807 kg	0.102 kg
1 kN	1/9.807 ton	0.102 ton
1 Nm	1/9.807 kg-m	0.102 kg-m
1 bar	1.020 kg/cm ²	

Cavity Pressure/Wall Thickness/Flow Path Length



If our product has a Flow Path Length of 150 mm and a minimum wall thickness of 0.8 mm, then the Flow Path to Wall Thickness ratio is $150/0.8 = 187.5:1$. By using the Cavity Pressure/Wall Thickness/Flow Path Length Chart above, we can determine that the Cavity Pressure with a minimum wall thickness of 0.8 mm is around 475 bar. The unit conversions tell us that 1 bar = 1.02 kg/cm², so:

$$\text{Clamp Force} = 475 \times 1.02 \times 120 = 58,140 \text{ or } 58 \text{ tons}$$

So that confirms it, we need the 60 ton machine for this particular job.

What happens if we change the material of our product? This brings us to yet another important factor – the viscosity of the polymer being moulded. Luckily for us, the viscosity of PS is 1.0, so all our calculations are right so far. But other materials have different viscosity factors, as can be seen in the table below and these must be taken into account when specifying an Injection Moulding Machine for our product.

Thermoplastic	Viscosity Factor
PS	1.0
PP	1.0 – 1.2
PE	1.0 – 1.3
ABS	1.3 – 1.5
PMMA	1.5 – 1.7
PC	1.7 – 2.0
PVC	2.0

If we decide to make our product out of Polycarbonate instead of Polystyrene, then we will have to multiply our results by the viscosity factor for PC thus:

$$58,140 \times 2.0 = 116,280 = 116 \text{ tons}$$

In reality nowadays, the Clamping Force is determined using software, but the programmes use the same principles, constants and conversions as we've used here.

SIZE

MOULDS

The size of an Injection Moulding Machine is described in Europe using the EUROMAP (European Committee of Machinery Manufacturers for Plastics and Rubber Industries) size rating which uses the clamping force in kN, and the product of the injection pressure in kbar and the injection volume in cm³. The EUROMAP size rating for a machine would be worked out as follows:

Injecto 150 Specifications

Clamping Force		Conversion (1 ton = 9.807 kN)	
150 tons	x	9.807	= 1471 (A)
Injection Pressure		Injection Volume	
2,000 kg/cm ² = 1.96 kbar	x	300 cm ³	= 588 (B)
		EUROMAP size rating (A:B)	= 1471:588

GENERAL

Most injection moulds are made in two halves and when you look at most injection moulded products, you can often see a line, which traces the meeting point of the two halves. You can also sometimes see the point at which plastic was injected and the resultant sprue cut off – called a Gate Mark. Products are always designed so that the Gate Mark is concealed on the inside or the underside of a product.

The mould is as important to the process as the injection moulding machine itself and often costs as much. The main factors affecting a mould's price are:

- Choice of material
- Number of cavities
- Size
- Texture of surface finish
- Complexity of part

Sometimes approximate conversion factors are used, in which case the rating would be 1500:600. In Asia, the number order is reversed and the Clamping Force is described in tons rather than kN, so the same machine would be said to have a rating of 600:150.

MOULD OPENING STROKE

This is the distance that the moving mould half moves from mould closed to mould open. Because the injection moulded part has to clear the mould and have room to be removed from the machine, the Opening Stroke must be greater than:

$$(2 * \text{Mould Height}) + \text{Length of Sprue.}$$

MOULD HEIGHT

This should probably be called Mould Thickness, the expression is left over from the days when moulding machines were vertical. Injection Moulding machines are normally adjustable and can accommodate a range of Mould Heights – expressed in the machine specifications as the Minimum and Maximum Mould Height.

THE MECHANICS

Normally, one half of the mould – the one next to the nozzle – is stationary while the other half moves for clamping and product release. At the end of a cycle, the Moving Mould Half moves back, pulling the product out of the Stationary (or 'Hot') Mould Half. An ejection system is built in at the end of the platen stroke to push the product off the Moving Mould Half so that it can easily be taken off the machine either manually or by an automatic collection system.

HEATING/COOLING

Because thermoplastic products have to cool before the mould can be opened, water or oil cooling systems are incorporated in the mould design, to reduce the cycle time. Because thermoset products sometimes have to be heated to allow for curing, those moulds may have to incorporate heating and cooling systems.

MATERIALS

Moulds are usually made from stainless steel or hardened tool steel or a combination of both. Occasionally moulds are made from aluminium.

DESIGN

The key to successful Injection Mould design is ensuring there are no undercuts, which can prevent the mould halves from opening without destroying the part. It is also useful if the design is shaped so that when the Moving Mould Half moves back (when the mould opens) the product stays on it – and not the Stationary Mould Half. Then the product will be released by the Ejector System.

VARIATIONS

Having only two halves and one injection point limits the complexity of the shape that can be produced and there are a lot of variations on this basic theme, which produce more complex shapes and indeed, more complicated moulds.

Rotary table injection moulding, for example, has a number of different mould cavities mounted on the stationary half of the machine and a rotating mould with a number of identical cavities mounted on the moving platen. After the first injection the moving platen rotates, taking the part with it, so that on the second injection, a new element is added to the part in a different material.

STUDENT NOTES