

Copyright Warning & Restrictions

The copyright law of the United States (Title 17, United States Code) governs the making of photocopies or other reproductions of copyrighted material.

Under certain conditions specified in the law, libraries and archives are authorized to furnish a photocopy or other reproduction. One of these specified conditions is that the photocopy or reproduction is not to be “used for any purpose other than private study, scholarship, or research.” If a user makes a request for, or later uses, a photocopy or reproduction for purposes in excess of “fair use” that user may be liable for copyright infringement,

This institution reserves the right to refuse to accept a copying order if, in its judgment, fulfillment of the order would involve violation of copyright law.

Please Note: The author retains the copyright while the New Jersey Institute of Technology reserves the right to distribute this thesis or dissertation

Printing note: If you do not wish to print this page, then select “Pages from: first page # to: last page #” on the print dialog screen

The Van Houten library has removed some of the personal information and all signatures from the approval page and biographical sketches of theses and dissertations in order to protect the identity of NJIT graduates and faculty.

ABSTRACT

Trouble Shooting in Plastic Injection Molding Machines

**by
Zafar Kamal**

The purpose of this thesis is to find the solution of those problems which occur during plastic injection molding process. Molding cycle problem analysis is an important field in plastic injection process. It is very necessary to catch the problems during operation. Therefore on the basis of research trouble shooting criteria is prepared.

Good quality control is an essential feature for finding any sort of fault. Quality control is associated with each and every step of operation to maintain the required shape and surface finishing.

Most of the molding problems are solved by varying the machine conditions and by changing the design of the mold. But some problems remain unchanged, therefore in such cases possible solution may be find out by means of examining the entire operation or by changing the variables. Therefore on the basis of this research a trouble shooting program or a program solver is prepared, which helps to get the correct solution of the problems.

TROUBLE SHOOTING IN PLASTIC INJECTION MOLDING MACHINES

by
Zafar Kamal

A Thesis
Submitted to the Faculty of
New Jersey Institute of Technology
in Partial Fulfillment of the Requirements for the Degree of
Master of Science

Department of Manufacturing Engineering

January, 1993

Blank Page

APPROVAL PAGE

Trouble Shooting in Plastic Injection

Molding Machines

Zafar Kamal

~~Dr. Steve~~ Kotefski, Thesis Adviser
Assistant Professor and Program Coordinator of
Manufacturing Engineering Technology, NJIT

Dr. Raj S Sodhi, Committee Member
Associate Professor of Mechanical Engineering and
Director of Manufacturing Engineering Programs, NJIT

Dr. Geskin E, Committee Member
Associate Professor of Mechanical Engineering, NJIT

BIOGRAPHICAL SKETCH

Name: Zafar kamal

Degree: Master of Science in Manufacturing Engineering

Date: January, 1993

Undergraduate and Graduate Education:

- Master of Science in Manufacturing Engineering
New Jersey Institute of Technology, Newark, NJ, 1993
- Bachelor of Science in Mechanical Engineering
N.E.D University of Engineering and Technology, Karachi, 1986

Major: Manufacturing Engineering

This thesis is dedicated to
my parents

ACKNOWLEDGMENT

I would like to thank my adviser Dr. Steve Kotefski for his guidance and advice in this research. I also thank Mr. Mohammed Nawaz without whose help completion of this work would have been difficult.

I would like to thank my friend Asim Zubeiri for his kind help during this research.

Specially, I am grateful to my cousin Aisha Nizam, who morally supported me to finish this degree.

Finally, I would like to thank my family, for their help, understanding and patience during the years that I studied.

TABLE OF CONTENTS

Chapter	Page
1 PROCESSING OF PLASTIC.....	1
1.1 Introduction.....	1
1.2 Extrusion.....	1
1.3 Injection Molding.....	2
1.4 Details of the Process	3
1.5 Clamping system	6
1.6 Molds	6
1.6.1 Different Features of Molds	7
1.6.2 Runners	8
1.6.3 Sprues	9
1.6.4 Venting	9
1.6.5 Mold Temperature Control	10
1.7 Heat Capacity of the Material	10
1.8 External Heating of Injection Cylinder	13
2 MOLDING AND FINISHING	17
2.1 Molding Cycle	17
2.2 Mold Cooling	22
2.3 Part Ejection	24
2.4 Mold Release	25
2.5 Additives to the Plastics	26
2.6 Mold Shrinkage	27
2.6.1 Post Mold Shrinkage and Warping	29
2.7 Quality Control	30

3 CLAMP SYSTEM	32
3.1 Introduction	32
3.2 Press Platen	32
3.3 Rating of the System	33
3.4 Efficiency of the Clamp	36
3.5 Opening and Closing System for Mold	36
4 SYSTEMS FOR INJECTION MOLDING MACHINE	38
4.1 Hydraulic System	38
4.2 Hydraulic Pumps	38
4.2.1 Operating Valves and Control	40
4.2.2 Hydraulic Motors and Cylinders	40
4.3 Hydraulic Fluids	41
4.4 Hydraulic Accessories	41
4.5 The Electrical System	41
4.6 Electrical Controls	42
4.7 Functional Controls	44
5 INJECTION MOLDING TECHNIQUES	46
5.1 Blow Molding	46
5.2 Extrusion Blow Molding	47
5.3 Injection Blow Molding	49
5.4 Stretch Blow Molding	52
5.5 Reaction Injection Molding	53
5.6 Liquid Injection Molding	57
5.7 Structural Foam Injection Molding	57
5.8 Sandwich Molding	60
5.9 Thermoforming	61
5.9.1 Vacuum Forming	61

6 FEED SYSTEM	65
6.1 Introduction	65
6.2 Runner.....	65
6.2.1 Runner Cross-Section Shape	66
6.3 Runner Size	70
6.4 Runner Layout	72
6.4.1 Single Impression Mold	72
6.4.2 Two Impressions Mold	74
6.4.3 Three Impressions Mold	74
6.5 Other Multi-Impression Molds	77
7 DESIGN OF THE GATES	78
7.1 Introduction	78
7.2 Location of the Gate	79
7.3 Balanced Gating	82
7.4 Types of Gates	83
7.4.1 Sprue Gate	83
7.4.2 Rectangular Edge Gate	84
7.4.3 Gate Size	87
7.4.4 Overlap Gate	88
7.4.5 Fan Gate	90
7.4.6 Tab Gate	91
7.4.7 Diaphragm Gate	92
7.4.8 Ring Gate	94
7.4.9 Film Gate	95
7.4.10 Pin Gate	95
7.4.11 Subsurface Gate	98

8 PLASTIC MOLDING MATERIALS	101
8.1 Introduction	101
8.2 Types of Plastics	102
8.3 Definition for Plastics	105
8.4 Identification of Plastic Materials	106
9 POLYETHYLENE AND POLYPROPYLENES	113
9.1 Low Density Polyethylenes	113
9.2 Injection Molding Polyethylene Lids	113
9.2.1 Molding Machines	114
9.2.2 Molding Conditions	116
9.2.3 Melt Temperature	116
9.2.4 Mold Temperature	116
9.2.5 Cycle Time	117
9.2.6 Injection Pressure and Injection Speed	117
9.2.7 Shot Size	117
9.2.8 Screw Speed	118
9.2.9 Materials	119
9.3 Polypropylenes	119
9.3.1 Physical Characteristics	123
9.3.2 Injection Molding Machines	123
9.3.3 Molding Thin Sections	126
9.3.4 Molding Thick Sections	126
9.4 Molding Techniques and Conditions Affecting Part	127
9.4.1 Injection Speed	127
9.4.2 Injection Pressure	128
9.4.3 Clamping Pressure	128

9.4.4 Melt Temperature	129
9.4.5 Mold Temperature	129
9.4.6 Cycle Time	130
10 CONTROL OF PROCESS	131
10.1 Introduction	131
10.2 Checking of the Material.....	131
10.3 Compounding and Coloring In-Plant	132
10.4 Production Control of Quality	134
10.5 Economic Control of Equipment	135
10.6 Machine Save Energy	137
10.7 Plastics Save Energy	141
10.8 Machine Safety	143
11 DIE TOLERANCES FOR PLASTIC MOLDING	144
11.1 Mold Shrinkage	144
11.2 Categories for Shrinkage Problems	145
12 TROUBLE SHOOTING PROGRAM	149
12.1 Introduction	149
12.2 Trouble Shooting	149
12.3 Approach Should be Exact	150
12.4 Problem Solving	151
12.5 Conclusion	152
REFERENCES	

LIST OF TABLES

Table	Page
1. Heat Required or Removed in Molding Typical Resins.....	11
2. Cylinder-Heater Inputs and Power Requirements for Typical Machine Size.....	15
3. Approximate Clamp Forces for Injection Shot Capacities.....	34
4. Typical Specification for a High Quality Turbine Oil Used as a Hydraulic Fluid.....	43
5. ASTM D 4000 Line Call-Out.....	107
6. Standard Symbols and Family Names.....	109
7. Cell Table G Detail Requirements.....	110
8. Suffix Symbols and Requirements.....	111
9. Tenite Polyethylene 1870 Physical Properties.....	120
10. Tenite Polyethylene 18B0 Physical Properties.....	121
11. Tenite Polyethylene 18D0 Physical Properties.....	122
12. Tenite Polypropylene 4E31 Physical Properties.....	125
13. Energy Requirement for Different Materials.....	142
14. Standard Tolerances Chart for a Polycarbonate.....	146
15. Standard Tolerance Chart for Phenolic.....	147

LIST OF FIGURES

Figure	Page
1. Cylinder Capacity as a Function of Melt Temperature	14
2. Injection Molding Cycle-Melt Ready for Injection	19
3. Injection Molding Cycle-Melt Being Injected	20
4. Injection Molding Cycle-Melt Being Plasticized	21
5. Injection Molding Cycle-Part Ejection	23
6. Schematic Diagram of a Basic Hydraulic Clamp System.....	35
7. Extrusion Blow Molding	50
8. Injection Blow Molding	51
9. Liquid Injection Molding	58
10. Stages in Liquid Injection Molding	62
11. Vacuum Forming	62
12. Feed System	67
13. Efficiency of Various Runner Profiles	69
14. Guide to Runner Diameter for Molding Section	73
15. Guide to Runner Diameter for Molding Section	75
16. Balanced Runner Layouts	76
17. Position of Gate	80
18. Runners Layouts Which Necessitate Balanced Gating	81
19. Sprue Gate.....	85
20. Different Shapes of Gates	86
21. Different Views of Overlap Gate	89
22. Different Views of Fan Gate	89
23. Different Views of Ring gate	93
24. Different Views of Tab gate	93

25. Different Views of Diaphragm Gate	96
26. Different Views of Position of Gate	96
27. Different Views of Pin Gate	99
28. Different Views of Round Edge Gate	99

CHAPTER 1

PROCESSING OF PLASTICS

1.1 Introduction

One of the most outstanding features of the plastics is the ease with which they can be processed. In some cases semi-finished articles such as sheets or rods are produced and subsequently fabricated into shape using conventional methods such as welding or machining. In the majority of the cases, however, the finished article, which may be quite complex in shape, is produced in a single operation. The processing stages of heating, shaping and cooling may be continuous (e.g. production of pipe by extrusion), or a repeated cycle of events (e.g. production of telephone housing by injection moulding), but in most cases the processes may be automated and so are particularly suitable for mass production.

There is wide range of processing methods which may be used for plastics. In most cases the choice of method is based on the shape of the component and whether it is thermoplastic or thermosetting. There are two principal methods for processing plastics.

1. Extrusion
2. Injection molding

1.2 Extrusion

One of the most common methods of processing plastics is Extrusion using a screw inside the barrel. The plastic, usually in the form of granules or powder, is fed from the hopper on to the screw. It is then conveyed along the barrel where it is heated by conduction from the barrel heaters and shear due to its movement along the length of the screw flights. The depth of the screw channel is reduced along the length of the screw so as to compact the material. At the end of the

extruder the melt passes through a die to produce an extrudate of the desired shape.

1.3 Injection Molding

One of the most common processing methods for plastics is injection moulding. Nowadays wide range of articles are made by means of injection moulding. These include such things as electric drill casing, television housings, gearwheels etc.

The original injection moulding machines are based on the pressure die casting techniques for casting metals. The first machine is reported to have been patented in the United States in 1872, specifically for use with celluloid. This was an important invention but probably before its time because in the following years very few developments in injection moulding processes were reported and it was not until the 1920s, in Germany, that a renewed interest was taken in the process. The first German machines were very simple pieces of equipment and relied totally on manual operation.

In the late 1930s the next major development in injection moulding, i.e. the introduction of the hydraulically operated machines, got available. However, these machines still tended to be hybrids based on the die casting technology and the design of injection moulding machines for plastic was not taken really seriously until the 1950s when a new generation of equipment was developed.

Basically, injection moulding is a simple process. A thermoplastic, in the form of granules or powder, passes from the feed hopper into the barrel where it is heated so that it becomes soft. It is then forced through a nozzle into the relatively cold mould which is clamped tightly closed. When the plastic has had sufficient time to become solid the mould opens, the article is ejected and the cycle is repeated. The major advantages of the process include its versatility in

moulding a wide range of products, the ease with which automation can be introduced, the possibility of high production rates and the manufacture of articles with close tolerances. The basic injection moulding concept can also be adapted for use with thermosetting materials.

1.4 Details of the Process

The earliest injection moulding machines were of the plunger type and there are still many of these machines in use today. A pre-determined quantity of moulding material drops from the feed hopper into the barrel. The plunger then conveys the material along the barrel where it is heated by the conduction from the external heaters. The material is then plasticised under pressure so that it may be forced through the nozzle into the mould cavity. In order to split up the mass of the material in the barrel and improve the heat transfer, a torpedo is fitted in the barrel. But there are few disadvantages in this type of machine i.e. it is difficult to produce consistent mouldings. The main problems are:

1. There is little mixing or homogenization of the molten plastic.
2. It is difficult to meter accurately the shot size. Since metering is on a volume basis, any variation in the density of the material will alter the shot weight.
3. Since the plunger is compressing material which is in a variety of forms the pressure at the nozzle can vary quite considerably from cycle to cycle.
4. The presence of the torpedo causes a significant pressure loss.
5. The flow properties of the melt are pressure sensitive and since the pressure is erratic, this amplifies the variability in mould filling.

Few of these disadvantages of the plunger machine may be overcome by using a pre-plasticising system. This type of machine has two barrels. Raw material is fed into the first barrel where an extruder screw or plunger plasticises the material and feeds it through a non-return valve into the other barrel. A

plunger in the second barrel then forces the melt through a nozzle and into the mould. In this system there is much better homogenization because the melt has to pass through the small opening connection the two barrels.

The shot size can also be metered more accurately since the volume of the material fed into the second barrel can be controlled by a limit switch on its plunger. Another advantage is that there is no longer a need for the torpedo on the main injection cylinder. But this type of machine is seldom used because it is considerably more complicated and more expensive than necessary.

Nowadays the market is dominated by the reciprocating screw type of the injection moulding machine. This was a major breakthrough in machine design and yet the principle is simple. An extruder type screw in a heated barrel performs a dual role. On the one hand it rotates in the normal way to transport, melt and pressurize the material in the barrel but it is also capable, whilst rotating, of moving forward like a plunger to inject melt into the mould.

There are a number of important features in reciprocating screw injection moulding machines and these will now be discussed here.

The screws in these machines are basically the same as those used in the extrusion. The compression ratios are usually in the range 2.5: to 4:1 and the most common L/D ratios are in the range 15 to 20. Some screws are capable of injecting the plastic at pressures up to 200MN/m. One important difference from a extruder screw is the presence of a back-flow check valve at the end of the screw. The purpose of this valve is to stop any back flow across the flights of the screw when it is acting as a plunger. When material is being conveyed forward by the rotation of the screw, the valve opens. One exception is when injection moulding heat-sensitive materials such as PVC. In such cases there is no check valve because this would provide sites where material could get clogged and would degrade.

The most recent development in the introduction of a vented barrel for injection moulding machines. It has the major advantage of permitting materials to be moulded without pre-drying.

The heaters are normally of the electrical resistance type and are thermostatically controlled using thermocouples. The nozzle is screwed into the end of the barrel and provides the means by which the melt can leave the barrel and enter the mould. It is also a region where the melt can be heated both by friction and conduction from a heater band before entering the relatively cold channels in the mould. Contact with the mould causes heat transfer from the nozzle and in cases where this is excessive it is advisable to withdraw the nozzle from the mould during the screw-backpart of the moulding cycle. Otherwise the plastic may freeze off in the nozzle.

There are several types of nozzle.

1. Open nozzle: The simplest is an open nozzle. This is used whenever possible because pressure drops can be minimized and there are no hold up points where the melt can stagnate and decompose. But in case of low viscosity of the melt then leakage occurs. The solution is to use a shut-off nozzle of which there are many types.
2. External shut-off nozzle: This is the nozzle which is shut-off by external means.
3. Needle shut-off nozzle: This is the nozzle with a spring loaded needle valve which opens when the melt pressure exceeds a certain value or alternatively when the nozzle is pressed up against the mould. Most of the shut-off nozzles have the disadvantage that they restrict the flow of the material and provide undesirable stagnation sites. For this reason they should not be used with heat sensitive materials such as PVC.

1.5 Clamping System

In order to keep the mould halves tightly closed when the melt is being injected under high pressure it is necessary to have a clamping system. Most common type of clamping systems are

1. Hydraulic
2. Mechanical

In the hydraulic systems, oil under pressure is introduced behind a piston connected to the moving platen of the machine. This causes the mould to close and the clamp force can be adjusted so that there is no leakage of molten plastic from the mould.

In the mechanical system, the toggle is the mechanical device used to amplify the force. Toggle mechanisms tend to be preferred for high speed machines and where the clamping force is relatively small. The main advantages of the toggle system are that it is more economical to run the small hydraulic cylinder and since the toggle is self locking it is not necessary to maintain the hydraulic pressure throughout the moulding cycle. The disadvantages of the system is that there is no indication of the clamping force and the additional moving parts increase maintenance costs.

1.6 Molds

An injection mould consists of two halves into which the impression of the part to be moulded is cut. The mating surfaces of the mould halves are accurately machined so that no leakage of the plastic can occur at split line. If leakage does occur the flash on the moulding is unsightly and expensive to remove. In order to facilitate mounting the mould in the machine and cooling and ejection of the moulding, several additions are made to the basic mould halves.

1. Backing plates permit the mould to be bolted on to the machine platens.

2. Channels are machined into the mould to allow the mould temperature to be controlled.

3. Ejector pins are included so that the moulded part can be freed from the mould.

The mould cavity is joined to the machine nozzle by means of the sprue. The sprue anchor pin then has the pulling the sprue away from the nozzle and ensuring that the moulded part remains on the moving half of the mould, when the mould opens. For multi-cavity moulds the impressions are joined to the sprue by runners. Channels cut in one or both halves of the mould through which the plastic will flow without restriction. A narrow constriction between the runner and the cavity allows the moulding to be easily separated from the runner and sprue. This constriction is called the gate. A production injection mould is a piece of high precision engineering manufactured to very close tolerances by skilled craftsmen. A typical mould can be considered to consist of

1. the cavity and core
2. the remainder of the mould.

Finishing and polishing the mould surfaces is also extremely important because the melt will tend to reproduce every detail on the surface of the mould. Finally the mould will have to be hardened to make it stand up to the treatment it receives in service. As a result of all the time and effort which goes into mould manufacture, it is sometimes found that a very complex mould costs more than the moulding machine on which it is used.

1.6.1 Different Features of Molds

Gates: The gate is a small orifice which connects the runner to the cavity. It has a number of functions. It provides a convenient weak link by which the moulding can be broken off from the runner system. In some moulds the degating may be automatic when the mould opens. The gate also acts like a valve in that it allows

molten plastic to fill the mould but being small it usually freezes off first. The cavity is thus sealed off from the runner system which prevents material being sucked out of the cavity during screw-back. Small gates are preferable because no finishing is required if the moulding is separated cleanly from the runner. So for the initial trials on a mould the gates are made as small as possible and are only opened up if there are mould filling problems.

In a multi-cavity mould it is not always possible to arrange for the runner length to each cavity to be the same. This means that cavities close to the sprue would be filled quickly whereas cavities remote from the sprue receive the melt later and at a reduced pressure. To get rid of this problem it is common to use small gates close to the sprue and progressively increase the dimension of the gates further along the runners. This has the effect of balancing the fill of the cavities. If a single cavity mould is multi-gated then here again it may be beneficial to balance the flow by using various gate sizes.

There are three types of gates which are commonly used.

1. Sprue gates: These are used when the sprue bush can feed directly into the mould cavity as in case of symmetrical moulding such as buckets.
2. Pin gates: These are particularly successful because they cause high shear rates which reduces the viscosity of the plastic and so the mould fills more easily.
3. Side gates: It is the most common type of the gate and is simple rectangular section feeding into the side of the cavity. A particular attraction of this type of gate is that mould filling can be improved by increasing the width of the gate but the freeze time is unaffected because the depth is unchanged.

1.6.2 Runners

The runner is the flow path by which the molten plastic travels from the sprue (i.e. the moulding machine) to the gates (i.e. the cavity). To prevent the runner

freezing off prematurely, its surface area should be small so as to minimize heat transfer to the mould. However, the cross sectional area of the runner should be large so that it presents little resistance to the flow of the plastic but not so large that the cycle time needs to be extended to allow the runner to solidify for ejection. A good indication of the efficiency of a runner is the ratio of its cross-sectional area to its surface area.

1.6.3 Sprues

The sprue is the channel along which the molten plastic first enters the mould. It delivers the melt from the nozzle to the runner system. The sprue is incorporated in a hardened steel bush which has a seat designed to provide a good seal with the nozzle.

Since it is important that the sprue is pulled out when the mould opens and there is a sprue pulling device mounted directly opposite the sprue entry. This can take many forms but typically it would be an undercut or reversed taper to provide a key for the plastic on the moving half of the mould. Since the sprue, like the runner system, is effectively waste it should not be made excessively long.

1.6.4 Venting

Before the plastic melt is injected, the cavity in the closed mould contains air. When the melt enters the mould, if the air cannot escape it become compressed. At worst this may effect the filling, but in any case the sudden compression of the air causes heating. This may be sufficient to burn the plastic and the mould surface at the local hot spots. To alleviate this problem, vents are machined into the mating surfaces of the mould to allow the air to escape.

1.6.5 Mold Temperature Control

If we want to get efficient moulding, we should control the mould temperature and this is done by means of passing fluid through a suitable arrangement of channel in the mould. The rate at which the moulding cools affect the total cycle time as well as the surface finish, tolerances, distortion and internal stresses of the moulded article. High mould temperature improves the surface gloss and tend to eliminate voids. If the mould temperature is too low then the material may freeze in the cavity before it is filled.

1.7 Heat Capacity of the Material

Thermodynamics principle are also valid for plastic materials, which are basic to the chemical structure of each. In the melting of various thermoplastics the quantity of heat required per unit weight may vary significantly depending upon the differences in the heat capacity of the various plastics.

Every material has a specific heat, which expresses its ability to absorb or release heat energy. Specific heat is the ratio of heat needed to elevate one pound of material one degree to that for heating water, which has a specific heat of 1, expressed as the number of BTU per pound per degree .

Due to complexity in handling specific heat data in calculation of heat input and removal, a mathematical extrapolation, heat content, may be used instead. The crystalline melting points for some materials can be seen by the change in the slope of the curves, making it obvious that a great quantity of heat is required to change the plastic's state from solid to liquid with the little change in temperature. The more crystalline materials require a large quantity of heat, the heat of fusion, to convert them from solid to liquid.

Typical examples of the total heat content of the plastics are available at any plastic injection molding book.

Table 1 Heat Required or Removed in Molding Typical Resins.

Resin	Average molding temperature, F.	Total heat added or removed Btu/pound	Heat of fusion, Btu/pound
General			
polystyrene	500	160	0
Polyethylene			
Low density	440	274	56
High density	440	310	104
Nylon	530	340	56

The plasticizing chamber or cylinder of the injection-molding machine is the primary element of the machine since it is responsible for the conditioning of the melt prior to its injection.

The size of the heating cylinder on a given injection-moulding machine is usually determined by the design of the parts to be moulded. Most machinery suppliers offer a range of the injection-unit sizes for machines of specific clamp-tonnage sizes. The determining consideration is the amount of the material needed to fill the mold which can have a large projected area with respect to part weight, requiring a large clamp tonnage.

Cylinders are rated as to their plasticizing rate to indicate how fast the melting system can soften a thermoplastic material to a flowable condition.

Thermoplastics vary, one from another, in many of the characteristics that determine their melting rates. Thus, for convenience the plasticizing rates of plasticators are usually related to one type of material as a standard, namely, general-purpose polystyrene. When an injection machine's shot or hourly heating capacity is given in ounces or pounds the reference to polystyrene is often omitted, but should be understood.

To calculate the corresponding plasticizing capacity of a heating cylinder for other resins it is necessary to know the relative specific heat of the materials. The capacity when using another resin is obtained by dividing its specific gravity by that of the polystyrene (1.04) and multiplying by the rated capacity. Thus for low density polyethylene (0.92 specific gravity) a cylinder rated at 20 ounces per shot will have a capacity of $0.92/1.04 \times 20 = 17.7$ ounces per shot.

The actual plasticizing capacity of a given heating cylinder depends to a large extent upon the plastic to be processed. The moulding temperature, thermal conductivity, specific gravity, and specific heat of a various materials are different and all of these factors play an important role in the complex processes of the

heat transfer and heat generating by shearing in the injection cylinder. Many of the newer thermoplastic materials, particularly those with lower thermal conductivities and higher melting points, require a careful analysis of the heating capabilities of the injection-moulding heating cylinders. When the density alone is considered for different thermoplastics over range of processing temperatures, the plasticising rates can be plotted for the various resins.

As production rate increases, the quantity of heat needed to achieve the proper melt viscosity also increases. The controlling and indicating instrumentation on most machines cannot reflect the actual temperatures within the barrel nor sense the true temperature within the plastic melt. The heater bands supply heat energy to the outside of the cylinder but effective heat input to the polymer will be less. Only a relative basis can be assumed for the settings of the instruments, since there is a lag between the indicated pyrometer readings and the actual stock temperature. And, different polymers will reflect a wider or narrower spread within these temperatures. The problem is further complicated by differences in the cylinder design, the depth of the thermocouples and the choice of the material as well as the rate of plasticization.

1.8 External Heating of Injection Cylinder

All injection-moulding machines make use of electrical heaters to aid in the plastification of the material being processed. The most common system uses mica heater bands of high wattage. Actually, several of these heaters are used per zone on the injection cylinder and, depending upon the size of the machine, the total wattage available for heating can be quite substantial.

Each zone along the length of the barrel is individually control by the thermocouple and a pyrometer. The thermocouple senses the temperature developed within the zone and transmit the information to the controlling

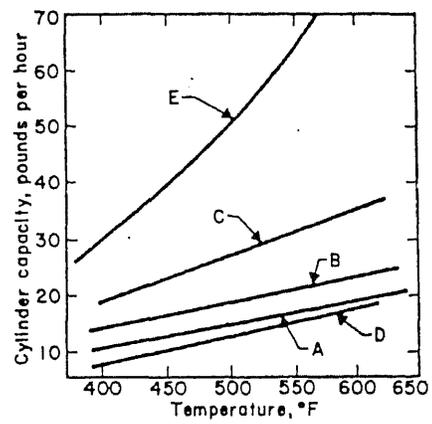


Figure 1 Cylinder Capacity
as a Function of Melt Temperature

Table 2 Cylinder-Heater Input and Power Requirements
for Typical Machine Size

Shot size Ounces	Plasticizing capacity, lbs per hour	Approximate heat input, kilowatts	Screw power input horse power
1	17.5	2.0----- 2.5	3.5
3	45	2.8----- 4.1	5
6	80	4.4----- 5.2	10
9	125	8.5----- 9.0	15
12	220	9.0----- 10.5	25
24	250	10.6----- 12.0	40
40	400	15 ----- 20	50
60	440	20 ----- 25	60
80	700	25 ----- 30	75
100	750	30 ----- 40	90
150	850	40 ----- 50	100
200	1000	60 ----- 70	125
225	1500	68 ----- 75	150
350	1800	75 -----100	200

pyrometer. When the temperature reaches the set point of the pyrometer, a signal is generated which tells the heater control to cut off the supply of electricity to that set of the heater bands. Likewise, when a given zone is indicated as being below the preset temperature level, a signal calling for a heat is sent back and the power is again applied to the heating bands.

There are a number of suitable instruments commonly used sense temperature along the barrel. The proportioning pyrometer is frequently found as standard equipment on most molding machines. Other types, which are usually more expensive, can be incorporated as optional equipment, when more accurate control is necessary.

To generate the proper signals for the pyrometer and control the heating zones, thermocouples are inserted into the barrel. A thermocouple works on the principle of generating a small electrical current within a bimetallic element as the temperature changes. The current is sensed by instrumentation within the pyrometer so that a signal is generated which can be calibrated to be read on a scale as a specific temperature. It is important to realize that the calibration for any instrument is affected by changes in the instrument itself, or in the choice of thermocouple, or in the length of the connecting wires with a resulting effect on the accuracy of the readings. Also, any damage to the thermocouple itself may result in erroneous readings.

CHAPTER 2

MOLDING AND FINISHING

2.1 Molding Cycle

The important thing to remember, while considering the cycle time is that, once the conditions are set into the controlling instruments of the machine, the identical cycle will be repeated accurately for the duration of the production run or until some effective change has been made.

In a typical cycle action is started by closing the safety gate at the die area. As it is closed the gate trips two safety limit switches, one for the machine hydraulic system, the other for the overall electrical system. From this point on, machine operation is automatic. Assuming that the machine is equipped with a hydraulic clamp, oil enters a jackram housed within the main ram, causing the movable platen to advance at high speed but under low pressure. Just before the mold halves make contact, a limit switch is tripped, causing oil to be diverted from the jackram to the main cylinder. Platen speed immediately drops off, but pressure increases. As the mold halves make contact, pressure builds up, sensed by the pressure switch that signals the injection unit to begin its cycle.

At this point, the reciprocating screw is in its retracted position behind a previously plasticized shot. Upon receipt of the signal from the pressure switch, the shutoff valve opens and the injector's hydraulic system is actuated. This causes the screw to advance as a plunger and to inject the shot at pressure as high as 20,000 psi. This initial pressure-high to counter the chilling effect of the mold is controlled by an adjustable relief valve or other flow control device. Length of screw stroke, and therefore shot size, is also variable and is controlled by a timer. When the mold cavities are filled, a signal from this timer to an injection-speed controller reduces pressure on the screw, permitting injection to

continue at a reduce rate. The purpose of maintaining melt pressure is to minimize the possibility of heat sinks resulting from contraction during part cooling. This phase of the cycle is called the dwell time.

At the end of the dwell time the timer signals the shutoff valve to close and the screw to resume its rotation. Plasticized melt again advances to the front of the screw, forcing it to retract against the back pressure until the shot-control limit switch is contacted. At this point, the clamp is still closed and the molded parts are cooling. On the signal from the timer, set according to the cooling-time requirements, the clamp opens slowly under pressure. Slow movement, mandatory to protect the molded parts, continues until a fast-return limit switch is contacted, at which point the platen speed picks up. Fast retraction continues until another limit switch is contacted by a control rod actuated by the platen, to reduce speed for slow, smooth ejection of the molded parts, which drop into a box, or water tank, or onto a conveyer. Thus the machine cycle and its subordinate, injection unit cycle are completed. In some cases automatic operation of an injection machine is interrupted for manual removal of parts that might be damaged by automatic ejection.

The overall molding cycle may be described as the total time required to produce one complete shot of one or more parts, depending upon the number of cavities in a given mold.

The molding cycle is not merely the time that the materials remains in the mold but includes the time necessary for the mold to close and clamp, any safety or delay time required at the start of the cycle, the injection time (time required to fill the cavity), the dwell or time required to cool the molten material, and the ejection time. Most molders refer to the sum of these elements as the gate-to-gate time or the total overall cycle.

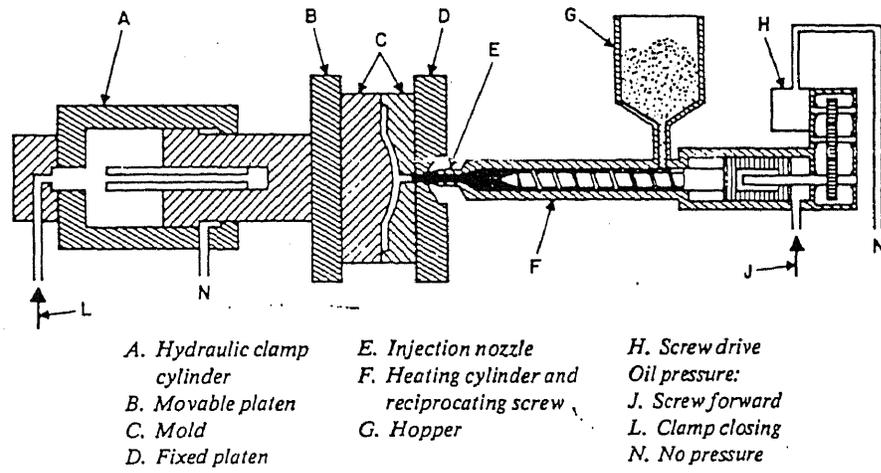


Figure 2 Injection Molding Cycle-Melt Ready for Injection

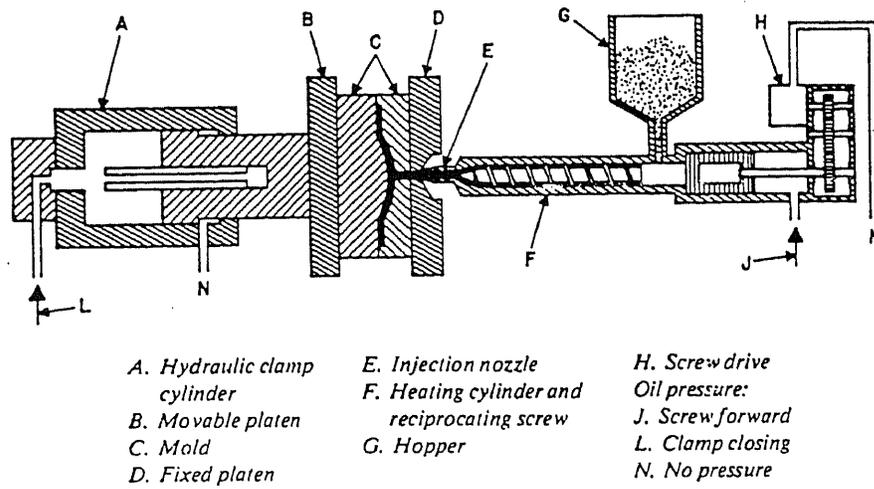
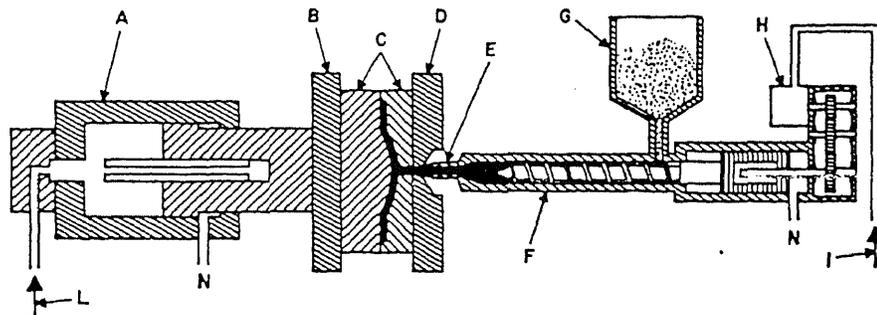


Figure 3 Injection Molding Cycle-Melt Being Injected



- | | | |
|------------------------------------|--|-------------------------|
| <i>A. Hydraulic clamp cylinder</i> | <i>E. Injection nozzle</i> | <i>H. Screw drive</i> |
| <i>B. Movable platen</i> | <i>F. Heating cylinder and reciprocating screw</i> | <i>Oil pressure:</i> |
| <i>C. Mold</i> | <i>G. Hopper</i> | <i>I. Screw drive</i> |
| <i>D. Fixed platen</i> | | <i>L. Clamp closing</i> |
| | | <i>N. No pressure</i> |

Figure 4 Injection Molding Cycle-Melt Being Plasticized Ejection

2.2 Mold Cooling

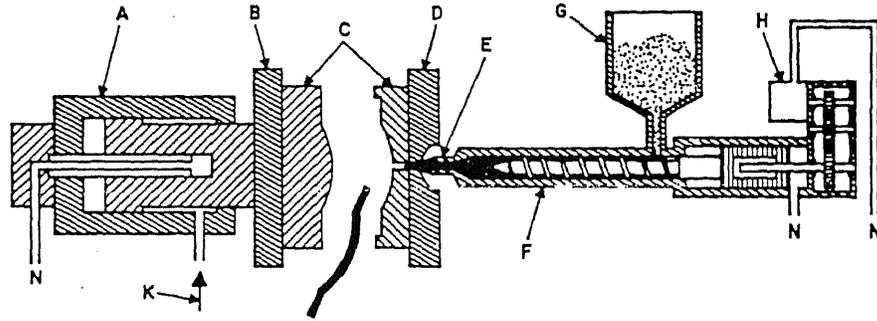
In any consideration of the injection molding process the subject of the mold cooling must not be overlooked. Obviously, if hot polymer is to set up within the mold cavity, the material must be cooled sufficiently to solidify it and allow the molded part to be removed from the mold.

This, in essence, is the mold-cooling portion of any cycle, and will vary in time and temperature depending upon the geometry of the part and the choice of thermoplastic material.

Mold cooling can be considered as taking place with the mold at any temperature below the inlet temperature of the plastic melt. Most commonly, the mold surface will be maintained at a temperature ranging from 30 to 40 degree F upto 120 to 150 degree F. The successful molder will strive to achieve as uniform temperature across the face of the mold as possible to develop within the mold a uniform shrinkage and thereby reduce the tendency to part warpage. When the injection mold is being designed and constructed, definite consideration must be given to the proper layout of the necessary cooling channels in both halves of the tool.

Remember that tool performance and the quality of the molded part depend largely on the ability to transfer heat rapidly and uniformly. In order to control shrinkage and warpage, it may be necessary to operate the mold at an elevated temperature level, with a somewhat longer cycle. When the molded part has a relatively thin wall section, and high speed production is a must, the mold may be operated using refrigerated cooling water. These are the extremes; normal molding operations will fall between these limits.

Obviously the heat necessary to effectively melt the plastic is the same quantity of heat that must be removed before the part can be ejected from the mold. As the heat content of the plastic increases and as the melt temperature



- | | | |
|------------------------------------|--|-------------------------|
| <i>A. Hydraulic clamp cylinder</i> | <i>D. Fixed platen</i> | <i>H. Screw drive.</i> |
| <i>B. Movable platen</i> | <i>E. Injection nozzle</i> | <i>Oil pressure:</i> |
| <i>C. Mold</i> | <i>F. Heating cylinder and reciprocating screw</i> | <i>K. Clamp opening</i> |
| | <i>G. Hopper</i> | <i>N. No pressure</i> |

Figure 5 Injection Molding Cycle-Part Ejection

goes up, it becomes even more important that the cooling system be capable of removing this greater amount of heat rapidly and efficiently. The way a mold is cooled is of great importance to the injection molder. Because lower mold temperatures make for increased part stiffness and strength, improved clarity, minimum shrinkage, and substantially reduce cycle times.

Molds are usually water cooled. But there is a limit to the number of cooling channels that can be incorporated into a mold base. These channels should be positioned so that the heat transfer from the melt is as uniform as possible. To obtain balanced heat transfer, greater cooling must often be provided around the thicker parts of the molded item.

Minimum warpage calls for high mold temperature; it also requires that the entire piece be cooled at nearly the same rate. This, in turn, calls for low temperature around the cavity gates and less cooling at the mold sections farthest from the gate area. To obtain fast cycle, it is important that the hottest portion of the mold, adjacent to the gating or sprue, receive the most concentrated cooling to minimize the total time required for the part to cool in the mold.

Having the mold too cold is risky for two reasons: It may result in short shots, and may yield molded parts with built-in stresses which can result in later stress cracking of parts.

2.3 Part Ejection

A very important step in the construction of the injection mold is part ejection. It depends upon the geometry of the part and the material from which it may be molded, and a number of other factors, the ejection system can vary in design and the method of the operation.

The method which is commonly used is knockout pin method which contact the molded part at its edges, or a flat area, or both, and are mechanically operated when the mold opens to eject the part. As the part becomes more detailed, the may be limited to contacting the runner system and not the part itself. Sometimes pins have blades or bar to provide more contact area, or the part may be lifted from the core by means of the stripper rings.

Some molded part requires very finish outlook therefore, air blast is used to remove the molded part from the cavity or core section of the mold. It is also important to consider the type of mold finish being used or may be tolerated, as an assist to proper part ejection. Jobs which are running at the high speeds or high production rates may employ some combination of these available methods. Part design, mold design, mold finish, cycle conditions, and mechanical ejection devices may all be incorporated within a given molding system to allow the job to run at the rates specified and to reasonably assure the continued operation of the machine without fear of parts hanging up in the mold. Obviously, any mechanical movement for actuating knockout devices, must also include a system for the resetting of the pin plate or return of the pins prior to the next shot, usually accomplished by use of knock-out-plate return pins. They are round pins slightly longer than the knockout stroke which contact the mold base on the closing prior to any interference of the pins with the cavity or core members.

2.4 Mold Release

Mold release is affected by a number of factors. Some molding compounds show better mold release than others, but it has been found that these compounds frequently have other disadvantages, such as greater shrinkage and less gloss. It is easy to understand what gloss has to do with the mold release, resins which

develop the grainy or frosty surface will release from the mold better than smooth, high-gloss resins. However, even similar materials may vary in mold release properties.

This problem may be alleviated by changing the mold design, or one or more of the molding conditions, without effecting the end properties of the molded part. Mold release may be affected if the mold is packed too tightly in an effort to reduce or control shrinkage. Also , a molded piece may stick if the injection time is too long and the part has shrunk on the core, a problem may be alleviated by reducing the injection time. On the other hand, the cycle may be too short to allow the molded part to shrink away from the cavity walls, therefore the time for the injection should be lengthened.

Mold release largely depend on the degree of the polish of the inside faces of the mold. Proper surface finishes within a mold for a deep-draw item will decide whether it can be ejected easily or will stick to the cavity or core half of the mold. Enough draft must always be provided, especially in deep draw articles. Reverse draft or relatively straight side walls should be avoided wherever possible.

2.5 Additives to the Plastics

A wide range of additives may be used into the plastic material before molding to enhance its end-use properties. The most common and the useful ingredient is pigmentation for the coloring of the plastic.

The better the color the better will be the sale of the plastic, it means salability rely heavily on color. The pigments can be incorporated in a number of ways. The greatest degree of pigment dispersion is accomplished through compounding, which is performed in separate facilities and is usually the most expensive method. When compounded colors are specified, inventory of the

plastic becomes a major concern, especially when a number of colors may be included in the production schedule.

The most common and the most widely used method is dry coloring, a procedure carried out completely within the molder's plant. Finely divided particles of pigment are tumbled with plastic material and adhere to the surface of the pellets through the static charge developed by most resins. In some instances, a coating or wetting agent may be used in conjunction with the dry pigment, such as the addition of a small amount of a nonreactive oil to the mix, to insure pigment adhesion to the plastic particles.

Dry coloring is the most economical system but does have some disadvantages. Since the pigment is only just adhere to the surface of the plastic care must be exercised in transferring the material to the hopper. Coloring can get into the surrounding air and contaminate other colors which may be used on other machines in the shop. Hoppers cover should be used on each press. The degree of dispersion with most plastics materials is not of the same quality as obtained by the other methods. The screw plasticator has greatly aided in dispersion.

Other ingredients are often added to the plastic such as antioxidants, stabilizers, lubricants, and the like. In some instances these can be added in the molding plant but the best results are achieved when these ingredients are compounded into the base resin or into concentrates,

Normally, these ingredients are used in small proportions and dispersing them is most difficult with the dry-trumbling technique.

2.6 Mold Shrinkage

Most nonplastic materials show relatively small changes in linear expansion and contraction under the influence of the temperature variation. On the other hand,

most plastics exhibits considerable dimensional change due to environmental changes.

Perhaps the most significant change in the plastics dimensions occurs with the phase change, causing the greatest concern to mold makers and the mold designers. It also represents a major problem to the injection molder. When going through the normal cycle of solid state to plastic melt and back to solid state plastics exhibit dimensional changes which vary widely from one resin to another. There is no practical way of accurately predicting the exact shrinkage allowance for any plastic material. A concerted effort has been expended during the past few years, particularly by the raw material suppliers to develop meaningful and useful data on the subject of the shrinkage. ASTM has set up a specification for determining mold shrinkage (D-955). But as with most laboratory determinations, the data developed has little or meaning to the molder since the parts he must produce rarely conform to the dimensions of the test bars or discs. Also, other variables such as gating, part geometry, variations in the wall thickness, and the efficiency of the machine itself, can make this data not only meaningless but also misleading.

Most plastics engineers must rely on their past experience both with the material under consideration and the type of part they are designing for, in order to closely approximate the shrinkage allowance to be made for a particular mold design.

A certain amount of shrinkage is inevitable in any process that involves cooling of a material from an elevated temperature. Mold shrinkage must be taken into account when a mold is designed by allowing for a calculated shrinkage based on the properties of the resins.

The injection molder can prevent excessive shrinking, which will make close tolerance impossible, by controlling the operation conditions, such as molding at

the lower injection temperatures, running a cold mold, or by packing the mold. Packing can be achieved by molding at either moderate melt temperatures and high melt pressures. However, excessive temperature or pressure may cause a mold to flash. Another means of reducing mold shrinkage is through the use of the high melt pressure and extended injection time, allowing additional resin to flow into the mold as the material in the mold cools and shrinks, again packing the mold as much as possible, short of causing the mold to flash or the part to stick in the cavity.

2.6.1 Post Mold Shrinkage and Warping

Basically the shrinkage in the mold is due to the thermal changes in the plastic while it is in the mold. The Post-mold shrinkage refers to the contraction of the solidified and ejected molded part due principally to relaxation of molded-in stresses, which is generally completed within one day after part ejection.

Although in many injection-molded parts shrinkage does not play too important a role, in certain applications it can present a troublesome problem for the molder. In closures and in pieces which are to become components of an assembly, too much or too little shrinkage can be a valid cause for the rejection of the part. In addition, uneven shrinkage causes warpage of the molded items.

Nonuniform shrinkage of a molded part after it is ejected from the mold, its bending or twisting out of shape, alters not only its dimensions but also its contours and angles. Warpage occurs mainly in large and the flat molded articles and, though undesirable in any molding, is particularly so in such items as container covers, closures, or drainboards.

When a part is ejected from the mold , it assumes its natural shape by relieving the stresses imposed upon it while being shaped in the mold in the viscous state. The problem for the molder and also a difficult one is to minimized

the internal stresses which the part may later remember, and relieve them when cooling it to room temperature. The locked-in stresses are generated in the mold by such operating conditions as excessive molding pressures, uneven mold cooling, or too low a melt temperature.

There is no single, clear-cut remedy for the warpage, the internal stresses set up in the molded part during in-mold cooling may be reduced by adjusting mold conditions, redesigning the part of the mold, switching to another resin, or some combination to these steps. Generally the best resistance to warpage results from maximum melt temperature, high mold temperature, minimum injection pressure, and short injection time.

Molding at high melt temperature tends to diminish the elastic memory of the resin and thus reduce the tendency to create the stresses that might cause warping. Running a warm mold allows stresses to relieve themselves before the melt sets or freezes, and will also reduce the tendency to warpage. In addition, uniform mold temperatures are a must to produce warp-free parts.

Part design and mold design have much to do with the warping. Warpage may be increased for instance, if the part has greatly dissimilar wall sections, if the gate is located in a thin section of the part, if the sprue is poorly placed, or if the mold is build up of inner surfaces with unequal heat dissipation.

2.7 Quality Control

Each company has a different interpretation of the need for and degree of quality control. Nevertheless, some measure of part usefulness must be made. In one instance it may be quick, visual inspection of the part for appearance only, while on the other hand, laboratory testing may be required to determine dimensional tolerance and other physical properties. Any inspection will add to the cost of the final part but this expenditure will also insure the usefulness of the product and its

acceptance by endues customer. It should be considered as part of the cost of doing business.

Simple inspection and gage testing is often done right at the molding machine, making the operator directly responsible for the quality of the parts he is producing. where quality control laboratories or inspection facilities exist, it may be necessary to hold all production until it is approved by these facilities before the molded parts may be delivered into inventory or shipped.

Because of the nature of some thermoplastics materials, some properties may not be developed to their maximum for several hours or even days. Where a fault is uncovered after such a time lapse, a considerable amount of production could be affected. In such cases, laboratories often developed a speedy test or an extrapolation based on the previous data to determine the acceptance of the production. However, this approach can only point to possible problems within the molding cycle or performance of the operation. Final acceptance would still be based upon the results of the completed testing program.

CHAPTER 3

CLAMP SYSTEM

3.1 Introduction

The function of the clamp of the injection molding machine is twofold. The fixed one is called platen and the other one is movable support the two halves of the injection mold and open and close the mold at the appropriate time in the molding cycle. The clamp unit also takes up the pressure applied to the plastic being injected by the injection unit during the injection part of the molding cycle.

To oppose the injection pressure necessary to deliver the highly viscous melt to the mold requires a significant amount of force. Without adequate force, the mold would simply open at its parting line and allow the molten material to escape. The result can be a reject part, or a mess which would require down time for clean up, and even more important, it could result in personal injury to the operator or other person in the vicinity.

3.2 Press platen

The platens of a molding press are heavy steel plates to which the halves of the mold are attached. In most machine operations, one platen is rigidly mounted and is correspondingly called the stationary platen. The other is mounted so that it can be moved as the clamp mechanism is opened and closed. The molds are attached to these plates either by direct bolting or through the use of clamps which are attached to the plates in a series of carefully spaced, drilled and tapped holes. This method provides a great degree of versatility to any given size machine. The stationary platen must provide an entry for the nozzle of the plasticizing chamber. A register ring or location device is provided in this plate to properly align the mold with the nozzle.

The movable platen must incorporate a device for actuating the part ejection system, most commonly by means of some mechanical knockout bar extensions which pass through the platen and contact the knockout plate built into the mold base. Some machines are equipped with hydraulic knockout arrangements whose actuating cylinders are mounted within the confines of the stationary platen.

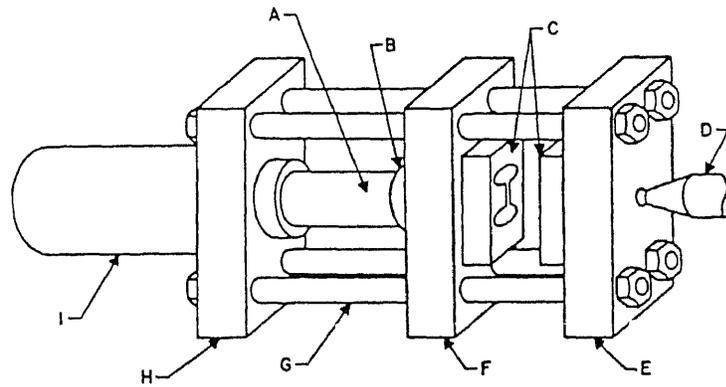
Because of the tremendous forces that may be applied during the clamping and injection portion of the cycle, the mold platens are ruggedly build. As the physical size of the press is increased, larger molds may be accommodated, and consequently the weight that may be suspended from the platen becomes greater. It should also be recognized that as the press size increases there is usually a significant increase in the clamp force that is available. The platen then, must not only be strong enough to support the tooling but also be capable of withstanding the bending or deflection action caused by the clamping force. Any serious distortion of the platen can cause the mold to flash and can lead to severe wear and damage to the press itself.

3.3 Rating of the System

The clamping system for the injection-molding machines have nominal ratings in tons of clamp force that can be exerted. Although machines are built to certain standards to provide a given force, individual units can vary somewhat from their design ratings. A large opposing force will be required to keep the mold closed and thus produce acceptable molded parts. As the part depth becomes greater and as the wall thickness becomes less, the opposing clamp force will become greater. Although the injection pressure in a given system may be in the range of 15000 to 20000 psi.

Table 3 Approximate Clamp Forces for Injection Shot Capacities

Clamp force tons	Shot size ounces
10 --- 25	Upto 2
25 --- 50	1 --- 4
50 --- 100	2 --- 10
100 --- 150	3 --- 15
150 --- 200	4 --- 25
200 --- 250	6 --- 30
250 --- 300	8 --- 40
300 --- 350	10 --- 50
350 --- 400	20 --- 60
400 --- 450	25 --- 70
450 --- 500	30 --- 80
500 --- 600	40 ---100
600 --- 700	50 ---120
700 --- 800	60 ---140
800 --- 900	70 ---180
900 ---1000	80 ---200
1000 ---1200	100 ---300
1200 ---1500	120 ---400
1500 ---2000	150 ---600
2000 ---4000	200 ---900



A. Actuating Plunger B. Removable Spacer C. Mold
D. Injection Nozzle E. Fixed Platen F. Movable Platen
G. Tie Bar H. Cylinder Base Plate I. Clamping Cylinder

Figure 6 Schematic Diagram of a Basic Hydraulic Clamp System

Pressure drops in the cylinder, through the nozzle, and into the mold cause a loss in the total available pressure and consequently, the effective pressure within the cavity on the material may be only 25 to 50 percent of the original starting pressure. With screw-type plasticators injection pressures are reduced primarily because of the greater degree of homogenization of the melt.

3.4 Efficiency of the Clamp

The nominal clamp-force ratings are approached but rarely reached in actual molding machines. A particular unit may be designed for a specific strength factor but more than likely it will not have the exact force indicated.

Since, in the course of any design calculation, there are many add-on factors which will indicate the need for more clamp force than the theoretical value, any loss in efficiency can usually be neglected. Also, as the machine is used its efficiency will continue to fall off from its original rating. Therefore, any machine that has been in operation for any length of time should be considered as having a lower clamp force than when it was new. If calculations indicate that the clamp force of a machine may be taxed by a particular job, other avenues should be sought for the running of this job.

One approach can be redesign of the mold to require less clamp force by reducing the projected area. The other one might be the choice of a material grade with a lower flow rating or less resistance to flow. The final choice may mean choosing a larger press having a greater available clamp force.

3.5 Opening and Closing System for Mold

The function of the clamp end of an injection molding machine is opening and closing of the mold. At the same time it must have a means of keeping the mold

tightly closed during the injection cycle. This function is performed by a mechanical or hydraulic device for traversing the movable platen of the machine.

Mechanical devices for moving and locking the movable platen consist of toggle systems which can be single or multiple, mounted between the movable platen and the fixed tailstock of the clamp unit. With the toggle in the open or folded position the movable and stationary platens are separated and the mold is open. When the mold is closed the toggle arm form a continuous beam structure that can support the full clamp force of the machine.

Clamp systems can be also be direct hydraulically operated. Hydraulic clamp cylinders can develop large clamp forces but they are slow acting as compared with toggle mechanisms. Hydraulic clamp systems are usually found on the larger capacity machines where toggle linkages that can support comparable clamping pressure may become too large and cumbersome for practical purposes .In addition to the simple mechanical-toggle and hydraulic systems many of the newest machines offer combinations. Small, fast-acting toggles are used to actuate the platen-moving mechaniesm, coupled with the hydraulic cylinders which into play during the final clamping and the actual injection cycle to apply the full clamp pressure of the machine.

CHAPTER 4

SYSTEMS FOR INJECTION MOLDING MACHINE

4.1 Hydraulic System

The function of the hydraulic system in injection molding machine is to transmit the power from electric motor to the various moving parts of the machine, and to control the power. An electrical control system regulates the hydraulic system to control the direction, force, speed, and sequence of the machine cycle. The basic components of the hydraulic system are:

1-fluid reservoir

2-pumps

3-valves

4-cylinders

5-hydraulic motors

The hydraulic fluids transmits the power throughout the hydraulic system, and lubricates the pumps and valves. It is important to follow the recommendations of the machine manufacturer or the hydraulic-equipment supplier in selecting the fluid to be used.

The hydraulic lines form passageways in which the fluid from one component to the other in the circuit. The lines may be seamless steel tubes, hoses, or the hydraulic oil may flow through drilled manifolds. The reservoir is simply a storage tank for the hydraulic fluid. In addition, it helps to keep the fluid clean by allowing contaminants to settle out, and it also minimizes turbulence, and dissipates heat.

4.2 Hydraulic Pump

The pump pushes the fluid through the lines. The pumps most commonly used on injection-molding machines are the balanced-vane type. The pumping unit of

cartridge includes a cam ring, a slotted rotor, and vanes held between, with a wear plate and pressure plate at the sides. Pockets are formed by each two adjacent vanes, the ring rotor, and the side plates. As the pump rotates, the volume of each pocket become larger or smaller as the contour of the cam ring pushes the vanes in and out of the rotor slots. Ports in the side plates, connected to the pump inlet port, admit oil into the rotating cartridge to fill the pockets as they increase in size.

Other ports in the sideplates are connected to the pump discharge port and accept oil discharge from the rotating cartridge when the pocket decrease in volume. Two inlet ports are diametrically opposite each other, as are the two pressure (outlet) ports, to provide the hydraulic balance which eliminates all pressure-induced loads, resulting in longer pump life and less maintenance.

The pump can also be called rotary fluid motors since they impart rotary motion to a load. In general, they permit an extreme range of speed adjustments and have a good ratio of size and weight to horsepower. Fluid motors can be stalled out under load without damage.

Vane type pumps can operate at pressure in excess of 2000psi. Their maximum speed is generally about 2200 rpm, with the maximum output rating of 125 hp. External-spur-gear type pumps are available in a range of sizes, with peak operating pressure of 1500 psi, top speed of 2400 rpm, and maximum output of 50 hp.

Internal gear models are available with speed capabilities up to 3600 rpm. They generally develop up to 3 hp at 1200 psi. Piston-type hydraulic motors can generally develop up to 300 hp at oil pressure of 3000 psi, with some models to operate at pressure up to 5000 psi. Continuous operating speeds range up to a maximum of 5000 rpm.

4.2.1 Operating Valves and Control

The relief valve limits the maximum pressure in the hydraulic circuit. Since the pump is a positive displacement unit, it continuously pushes out oil regardless of the resistance which the flow encounters. The relief valve is needed to bypass excess oil back to the tank and protect the hydraulic system and machine against excessive pressure.

An unloading valve is used to dump the flow of oil from one pump, while holding pressure with another pump. This type of pump is generally used with a double pump to conserve power and heat by unloading the large-volume pump while holding pressure with the small-volume pump.

A check valve permits flow in one direction and prevents flow in the other direction. A light spring holds the poppet on its seat. When the pressure on the inlet port overcomes the spring force, the poppet lifts off its seat and flow passes from the inlet to the outlet port. In the reverse direction, pressure applied to the outlet port adds to the spring pressure, holding the poppet closed so there is no reverse flow. A flow-control and relief valve regulates the rate of flow and limits the maximum pressure in a hydraulic line. It is commonly used to control the speed of a hydraulic motor driving the screw on the injection-molding machines.

Solenoid-operated directional valves control the direction of the oil flow. They may be either solenoid-operated or solenoid-controlled and pilot-operated.

4.2.2 Hydraulic Motors and Cylinders

Hydraulic motors are used to convert hydraulic energy (pressure and flow) to mechanical energy. Their operation is reverse of a pump's. When oil is pushed into one port and discharge from another, the motor rotates in one direction.

If the flow is reversed, the direction of the rotation is reversed. Since no centrifugal force exists until the motor begins to rotate, springs are used behind

the vanes to hold them in place against the ring. Unlike motors , cylinders are linear actuators which convert hydraulic energy into mechanical energy. Oil forced in the one end of the cylinder causes the piston and rod to move in one direction. If the flow of oil is reversed the movement is reversed.

4.3 Hydraulic Fluids

If the heating cylinder of the molding machine can be treated as the heart of the system , then it follows the hydraulic-oil lines are the blood of the process. The selection of the oil is very important factor , therefore serious consideration should be taken while selecting the oil.

4.4 Hydraulic Accessories

All modern molding-machine are equipped with the necessary filtering systems, and a part of any good maintenance program includes the regular inspection of these systems. Maintaining the hydraulic fluid at the proper tank level, making sure that the injection machine is operated at the proper temperature levels, and assuring that the oil is being cleaned and filtered are all-important to the continued trouble-free operation of the machine. The other type of the system which is very important in case of accuracy achievement is called an electrical system.

4.4 The Electrical System

For better and accurate functions, the injection-molding machine follows a programmed sequence of events for the precise control of heat, the proper timing of injection pressures and cooling, and special machine sequences, and the ever-dominant need for safety of personnel and equipment. Most injection

that the operator has the fundamental knowledge of the machine's electrical circuits.

4.5 Electrical Controls

The electrical control system serves as the nerve and memory center to program and sequence the machine cycles. Its purpose is to sense, program the results, and cause an action to take place.

Position is sensed through the use of the limit switches, change in the heat through thermocouples, and changes in pressure through pressure switches. After analyzing the incoming signals and making a programmed decision, the control system passes this decision on to the pilot operating devices which convert the electrical signal into mechanical motions, thus resulting in the desired action.

There are specific type of the devices available for the different functions. First, There is motor or prime mover , and then the sensing devices, next, control systems such as motor starters, heater contractors and relays, and finally , the ultimate load controls such as solenoids or heater bands.

The electric motor or motors used in the average injection press must have a high breakdown torque. Most motors operate on 220 or 440 volts and are constant-speed types with the direct linkage to a pump or pumps. Some foreign equipment uses dual-speed motors, particularly where an electric drive is supplied for the rotation of the screw. Where electrical energy is the direct driving force for the screw system, speed-reduction gears are usually used so that the higher torque can be generated and the screw speed can be varied as needed for specific materials and molding applications. The use of electrical and mechanical energy is useful for every system and the combination is widely used.

Table 4 Typical Specification for a High Quality Turbine Oil Used as a Hydraulic Fluid

PROPERTY	Value
Viscosity index	89
Viscosity at 100 degree fahrenheit, S.S.U.	315-325
Pour point, S.S.U.	6000
Steam emulsion	125
Neutralization number	0.06

4.6 Functional Controls

Control devices take many forms, both in physical appearance and in their functional operation. The most widely used device is the relay. Regardless of the application, the purpose is always to switch a control circuit from one function to another. A relay is composed of a coil, a magnetic structure, an armature, and a set of contacts. When the current is applied to the coil, all contacts change from their normal to their energized switching position.

Contactors are the devices which are similar to relays in general operation. Their purpose is to switch circuits, except that unlike relays, they are designed to carry heavier loads or currents. Contactors are usually sized according to the current-carrying capacity of the contacts. Operationally there is no difference between a contactor and a relay. Magnetic starters are devices used to control the starting and the stopping of the motors. Basically, they are contactors with added features.

Solenoids are devices used to obtain a straight-line motion. They are available in push or pull types, or may be double-acting. The pull of a solenoid must at all times exceed the load, if it less, the solenoid action will be sluggish and may not complete the stroke. In the case of the solenoid-operated valve as used in injection-molding machines, such sluggish action would mean that the valve spool would not complete its stroke, thereby preventing oil from passing through its proper pressure or tank port.

With automatic machinery various types of signal or detection devices are used to cause the controls to properly program their respective functions. These devices are found in many forms, but their purpose is always to either signal a change in operation, present a warning, or indicate some other action. There are limit switches to sense and detect position or pressure, and vacuum switches

which sense high low pressures or vacuum levels, float switches which detect liquid levels, and temperature detectors to sense heat.

Timers are a form of control used in process programming to signal elapsed or delayed time of a cyclic sequence. The electronic timer is usually used for precise repetitive accuracy, especially in the small size range. Where time setting of 3 to 150 seconds are required, along with the external settings, fair repetitive accuracy, and where simple repair and maintenance is allowable, the synchronous-motor timer is the most economical. In the plastic industry, heat plays an important role. Along with heat there must be a good, accurate type of the temperature control. One of the most efficient control instrument is the pyrometer which uses a thermocouple or millivolt input. Voltages generated by the thermocouple operate the temperature-calibrated millivoltmeter, which indicates temperature, and bring into action various means for performing control functions. The electromotive force generated at the thermocouple tip is amplified and actuate the meter element to move an indicator pointer across the calibrated scale. If the pointer is below a set point, electronic action within the instrument energizes the relay which, through external circuitry, causes the heater to energize. When the pointer has reached the preset point, the relay deenergizes, causing the heaters to deenergize. It is through this on-and-off action that the pyrometer controls heat within narrow limits.

On most injection-molding machines, the barrel-heater bands are generally made in two halves and clamped to the cylinder with the strap or a clamp device. Each half heater has its own separate 220-volt winding.

CHAPTER 5

INJECTION MOLDING TECHNIQUES

5.1 Blow molding

Hollow plastics products such as a squeeze bottles, milk bottles, fuel tanks, toys, oil containers, chemical tanks, furniture, electrical housing, are blow moulded. Different processes are used but basically all of them are similar.

The basic process involves producing a plastic parison or preform (tube, pipe, or test tube plastic shape), placing this preform into a closed two plate mold, injection of air into the heated parison to blow it out against the mold cavity, cooling of the expanded parison, opening of the mold, and removing the rigid blow molded part.

Blow molding technique basically divided into three categories, namely, the extrusion blow molding process which principally uses as unsupported parison, and the injection blow molding process which uses a preform supported on a metal core pin. The third major category is called the stretch-blow molding process. Stretch-blow molding can be started with either the extrusion or injection blow molding process. By stretching at prescribed temperature the properties of many plastics can be significantly improved providing cost / performance advantages.

These processes provide different advantages to produce all type of products; so it is necessary to examine the process to be used based on product / performance requirements, material performance and the production quality. As an example , the plastic bottle does more than hold the product. It combines safety, light weight, design freedom, appealing colors, and low energy usage. Other factors to be examine in the blow container can be the desired shelf life, moisture barrier, oxygen barrier, drop strength, heat distortion, compatibility of

plastic and product, top load, environmental stress cracking, clarity requirements, coloring of the plastic material, and cost.

In extrusion blow molding the advantages include high rate of production, low tooling cost, blown handle ware, wide selection of machine manufacturer. Disadvantages are usually high scrap rate, recycling of the scrap, limited wall thickness control or material distribution, the fact that the trimming can be accomplished in the mold for certain type mold or secondary trimming operations have to be included in the production lines.

With injection blow molding, the major advantages include the fact that no scrap or flash is molded, best wall thickness and material distribution control, best surface finish of the part, low volume production quantities which are economically feasible. Disadvantages are high tooling cost, no handle ware, the fact that based on the cost to produce large extruded blow molded parts the injection blow molded is limited to smaller sizes. Advantages and disadvantages are similar for the stretched blow molding. The major advantage is that cost / performance can be significant to certain sizes of products such as the carbonated beverages bottle.

Important factors to be consider when examine the blow molded process to be used usually start with the part size, number to be manufacturer, design / shape, and cost limitations.

5.2 Extrusion Blow Molding

In extrusion blow molding, a parison is formed by an extruder. The plastic pellets are melted by heat which is transferred from the barrel and by the shearing action of the extruder screw as they pass through the extruder. The helical flights of the screw change configuration along its length from input to output ends to ensure a uniformly homogeneous melt.

Turning continuously the screw feeds the melt through the die head as an endless parison or into an accumulator. The size of the part and the amount of the material necessary to produce the part (shot size) dictate whether or not an accumulator is required. The non-accumulator machine offers an uninterrupted flow of the plastic melt.

With the accumulator the flow of the parison through the die is cyclic. The connecting channel between the extruder and the accumulator, and within the accumulator itself, are design rheologically to prevent restrictions that might impede the flow or cause the melt to hang up. Flow part should have low resistance to melt flow to avoid placing an unnecessary load on the extruder.

To ensure that the least heat history is developed during processing, the design of the accumulator should provide that the first material to enter the accumulator is the first to leave when the ram empties the chamber; and the chamber should be close to totally emptied on each stroke.

When the parison or tube exits the die and develop a preset length, a split cavity mold closes around the parison and pinches one end. Compressed air inflates the parison against the hollow blow mold surfaces, which cools the inflated parison to the blow mold configuration. Upon contact with the cool mold wall, the plastic cools and set the part shape. The mold opens, eject the part, and closes around the parison to repeat the cycle.

Various techniques are used to introduce air into the parison. It may be accomplished through the extrusion die mandrel, through a blown pin over which the end of the parison has dropped, through blown head applied to the mold, or through blowing needles that pierce the parison. The wall distribution and the thickness of the blown part are usually controlled by parison programming, blow ratio, and the part configuration.

The mold clamping methods are hydraulic and / or toggle actuation. Sufficient daylight in the mold platen area is required to accommodate parison system, unscrewing equipment.

Clamping system vary based on the part configuration. Basically there exist three types. The "L-shape" has the printing line at an angle of 90 degrees to the center line of the extruder. The "T-shape" has the parting line inline with the extruder center line. Mold opening is perpendicular to the machine center line. The third method is the gantry type. The extruder/die unit is arranged independently of the clamping unit. This arrangement permits the clamp to be positioned in either the "L" or "T" shape without being tied directly into the extruder.

The basic extrusion blow molding machine consists of an extruder, cross-head die, clamping arrangement, and mold. Variations include multiple extruders for co-extrusion of two or more materials, parison programmer to shape the parison to match complex blown part shapes, and the multiple station clamp systems to improve output through the use of the multiple molds.

5.3 Injection Blow Molding

In injection blow molding, a preform is formed by a conventional injection molding machine plasticator. The injection molding machine injector provides an optimum plastics melt, with the uniformly homogeneous melt that is repeatable. This plastic melt is injected into the preform cavity forming the preform around the core rod.

In tool design the core rod and the parison are very important. Each container to be blow molded has its unique parison and core rod design.

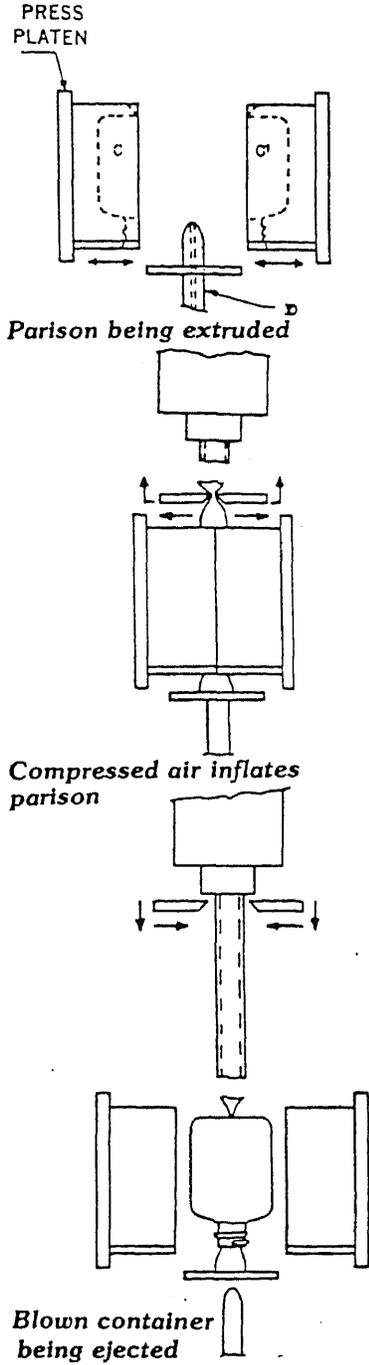


Figure 7 Extrusion Blow Molding

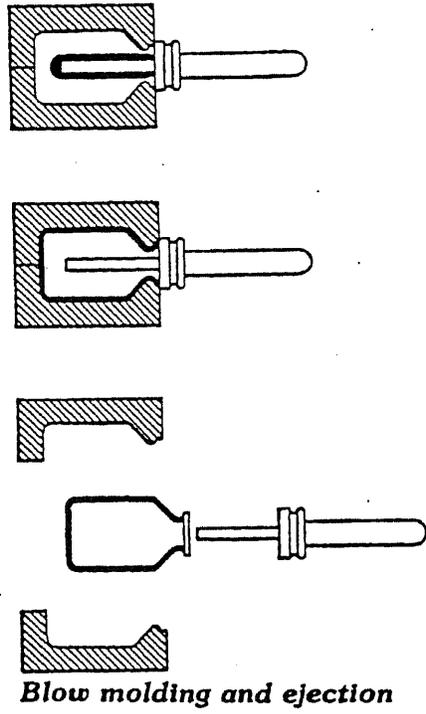
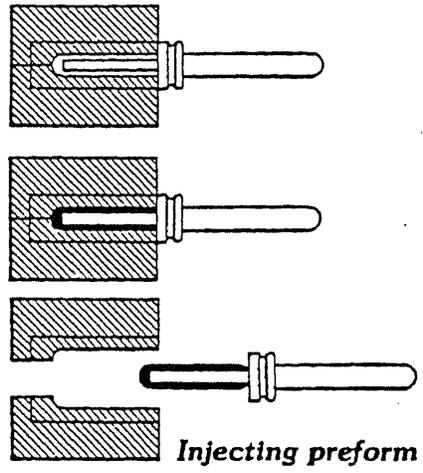


Figure 8 Injection Blow Molding

The second stage consists of transferring the injection molded preform, via either the core rod or the neck ring, into the blow mold. At this station compressed air enters through the core rod or seal ring, and the preform is blown into the blow mold configuration . It is held in the cold blow mold until the material is set, and then the air is exhausted and the blown bottle ejected.

Machines are used with from two to six stations. In the two-station machine, the finished container is ejected after the blow mold opens at the blow station by air pressure or by mechanical means. In the more conventional three-station machine, the finished container stays with the core pin as it is indexed to the third station where ejection take place. Four, five and six-station machines are available. These additional stations are used for further processing of the containers, such as decorating, position of the blown parts, filling. There have been several type of machines available with different methods of transporting the core rod from one station to other. These include the shuttle, two-position rotary, axial movement, and rotary with three or more stations used in the conventional injection molding clamping units.

5.4 Stretch Blow Molding

The stretch-blow process can give many resins improved physical and barrier properties. In biaxial orientation, bottles are stretched lengthwise by an external gripper, or by internal stretch rod, and then stretch radially by blow air to form the finished container against the mold walls.

This process aligns the molecules along two planes, providing additional strength and even more important, better barrier properties than are possible without biaxial orientation. Other advantages include better clarity, increase

impact strength, or toughness, and reduce creep. The actual increase is dependent on the ratio of blow-up in each direction.

Stretch blow molding is possible for thermoplastic materials such as PET, PVC, polystyrene, acrylonitrile, polypropylene, and acetals. The amorphous material with the wide range of thermoplasticity are easier to Stretch blow than the partially crystalline types. With the partially crystalline type, if the crystallizing is too rapid, the bottle is virtually destroyed.

Stretch-blow processing can be separated into two categories: in-line and two-stage. In-line processing is done on a single machine, while two stage processing requires an injection line to produce preforms, and a reheat blow machine to make the finished bottles. In the in-line, an injection molded parison passes through the conditioning stations that bring it to the proper orientation temperature. A rather tight temperature profile is held in the axial direction of the preform. Advantages of the in-line systems are that heat history is minimized, and the preform can be programmed for optimal material distribution if it is maintained under continuous control.

With the two-stage, the process uses extruded or injection molded preforms that have been cooled, and indexes them through an oven that reheats them to the proper orientation-blow temperature. Advantages of these processes can be the fact that scrap production is minimized, improved thread finis, higher output rates, and the capability to stockpile preforms.

5.5 Reaction Injection Molding

This process involves the high pressure impingement mixing of two or more reactive liquid components and injected into a closed mold at low pressure. With RIM technology, cycle time of 2 minutes and less have been achieved in production for molding large and thick parts. Principal plastic used is

thermopolyurethane (PUR). other materials used are thermoplastic nylon; thermoset polyester and epoxy.

The advantages of the RIM over injection molding include the molding of the parts larger than 10 pounds, they can be made on the production basis using thinner walls because of the lower processing viscosities, or using very thick walls because curing is uniform throughout the part. There are problems associated with this method, however. The lack of the suitable internal release has made the RIM process labor-intensive, but changes are now occurring to significantly reduce or eliminate this problem.

The molded polyurethane faithfully reproduces the surface of the mold and tends to stick to them. Originally the application of the mold-release agents was necessary with each cycle of the RIM technology. After polymerization, if the mold is not covered with the mold-release agent, the part will adhere to the mold, making it difficult to remove from the mold. In view of these occurrences, the mold material should be highly polishable and platable with nickel, this coating has proved to be the most effective in product removal.

In the processes of injection molding of thermoplastic, injection molded thermoset, structural foam molding, and expandable polystyrene molding, we are dealing with the materials which are chemically complete compounds, ready for conversion into finished part. The materials are received from the suppliers with the certain properties based on the test bar information and recorded in material processing data sheets. The processors are expected to convert these materials into the products with similar mechanical, electrical, and environmental characteristics, as indicated on the data sheets. The processors also furnished with the range of molding parameters that should be optimized to attain the desired production properties. In reaction injection molding, the starting point for the conversion process are liquid chemical components. These components are

metered out in proper ratio, mixed, and injected into a mold where the finished product is formed. In reality, it is a chemical and molding operation combined into one system of molding in which the raw material is not a prepared compound but chemical ingredients that will form a compound when molded into a finished part. The chemicals are highly catalyzed to induce extremely fast reaction rates. The materials that lend themselves to the process are urethane, epoxy, polyester, and others that can be formulated to meet the process requirement.

The system is composed of the following elements:

1. Chemical components that can be combined to produce the material of desired physical and environmental properties. Normally, this formulation consists of two liquid chemical components that have suitable additives and are supplied to the processors by chemical companies.
2. A chemical processing setup, which stores, meters, and mixes the components ready for introduction into the mold.
3. To facilitate smooth continuous operation, a molding arrangement consisting of a mold, mold-release application system, and stripping accessories.

The success of the operation will depend on the processor's knowledge of:

1. The chemistry of the two components and how to keep them in good working order.
2. How to keep the chemical adjunct in proper functioning condition so that the mixture entering the mold will produce the expected result.
3. Mold design as well as the application of auxiliary facilities that will bring about ease of the product removal and mold functioning within a reasonable cycle. RIM molding is energy saving as compared to the conventional injection molding. The two liquid urethane components are injected generally at room temperature, and atypical mold temperature is 150 degrees. Also, since the material is expanded after injection, very low clamp pressures (100 psi) are required.

Since internal mold pressures would not normally exceeds 100 psi, the clamping requirements for RIM are substantially lower than that for thermoplastic processing. Calculations have been done on the part and show that a clamp requirement of 2500 to 5000 tons necessary to produce a part from conventional injection molded thermoplastic polyurethane can be reduced to less than 100 tons for RIM.

The production of polyurethane elastomers involves the controlled polymerization of an isocyanate, a long- chain-backbone polyol and a shorted-chain extender or cross-linker. The reaction rate can be controlled through the use of the specific catalyst compounds, well known in the industry to provide sufficient time to pour or otherwise transfer the mix, and to cure the polymer sufficiently to allow handling of the freshly demolded part. The use of the blowing agent allows the formation of the definite cellular core as well as a nonporous skin, producing an integral sandwich-type cross section.

Reaction injection molding involves very accurate mixing and metering of two highly catalyst liquid urethane compounds, polyol and isocyanate. The polyol components contains the polyether backbone, a chain extender or cross-linking agent, and a catalyst. A blowing agent is generally included in either the polyol or isocyanate component. In order to achieve the optimal in physical properties and part appearance, instantaneous and homogeneous mixing is necessary. Insufficient mixing results either in the surface defects on the part or, at the time of the posture, delamination or blistering.

The urethane liquid components are stored at a constant temperature in a dry air or nitrogen environment. These components are delivered to high pressure metering pumps or cylinders that dispense the respective materials at high pressure and accurate rates to a mixing head. The material are mixed by the

stream impingement. Additional mixing is generally encouraged via a static mixer incorporated into the runner system of the mold. Following the injection of the chemicals, the blowing agent expands the material to fill the mold.

The preferred route for high-volume RIM manufacturing is multiple clamps fed from a single metering pumping unit, the logic being that this is the most efficient way to utilize the capacity of the mold filling equipment.

5.6 Liquid Injection Molding

The process of liquid injection molding has been used longer than RIM. From the practical view these two methods are similar. Their concept of automated low pressure processing of the liquid thermosets in converted injection machines has conclusively demonstrated advantages of faster cycles, lower labor rates, lower capital investment, energy saving and space savings relative to the conventional potting, encapsulation, compression transfer processes, and conventional injection molding.

A major application for the LIM-silicones continues to be encapsulating electrical / electronics devices. The usual LIM system basically uses two or more pumps that moves the components of the liquid system to a mixing head, before they are forced into the heated mold cavity in force. There are system in which screw mixing is used, similar to conventional injection molding.

5.7 Structural Foam Injection Molding

Foamed thermoplastic articles have a cellular core with relatively dense skin. The foam effect is achieved by the dispersion of the inert gas throughout the molten resin directly before the molding. Introduction of the gas is usually carried out by either pre-blending the resin with the chemical blowing agent which releases gas when heated or by direct injection of the gas.

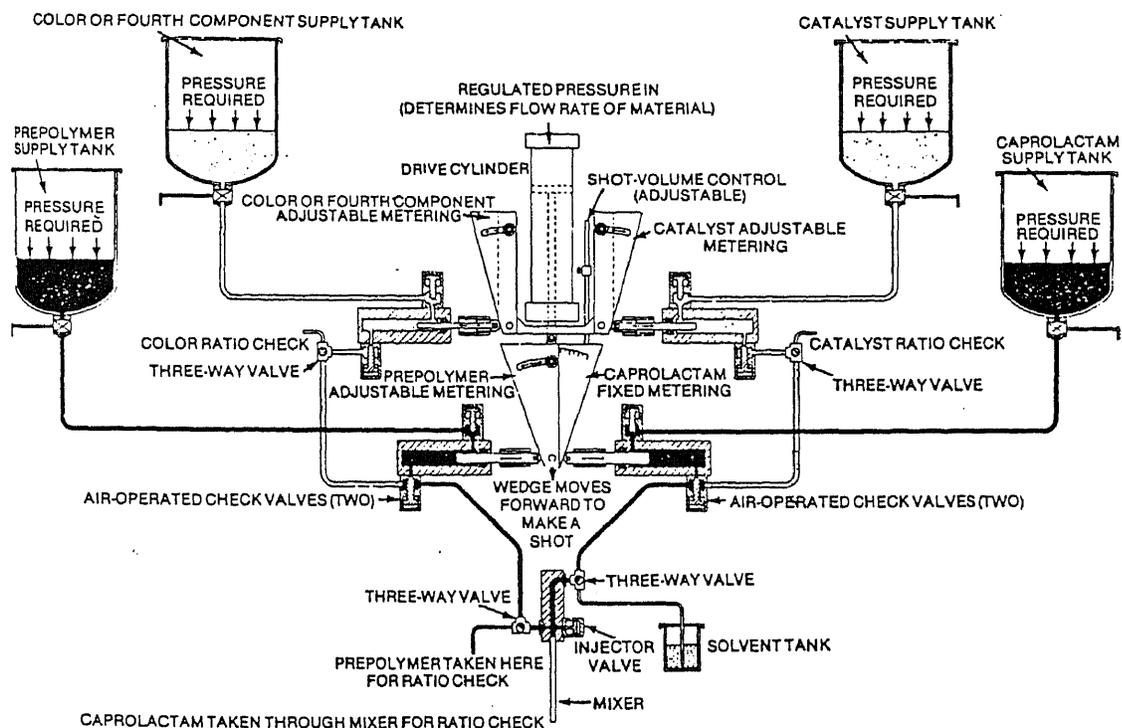


Figure 9 Liquid Injection molding

When the compressed gas / resin is rapidly injected into the mold cavity, the gas expands explosively and forces the material into all parts of the mold.

The advantages of these type of foam moldings are:

1. For a given weighed they are many times more rigid than a solid molding.
2. They are almost completely free from orientation effects and the shrinkage is uniform
3. Very thick section are molded without sink marks.

Foamed plastic articles may be produced with good results using normal screw-type injection molding machines. However, the limitation on the shot size, injection speed and the platen area imposed by the conventional injection equipment prevent the full large-part capabilities of a structural foam from being realized. Specialized foam molding machines currently in use can produce parts weighing in excess of 50 kg.

Wall sections in foam molding are thicker than in solid material. longer cycle times therefore be expected due to both the wall thickness and the low thermal conductivity of the cellular material. In contrast, however, the injection pressures in foam molding are low when compared with the conventional injection molding. This means that less clamping force is needed per unit area of molding and mold costs are less because lower strength mold materials may be used. A recent development in this field is a process of cinpress which an acronym for control injection pressure. In this process a gas, normally nitrogen, is injected into the plastic melt as it enters the mold. The conditions under which this occurs must be precisely controlled in order to produce laminar flow. The gas does not mix with the melt but forms the continues channel which, because the surface tension of the plastic, does not break through to the surface of the mold. The gas follows the path of the least resistance at the center of the melt path. The resulting

molding consists of "box-section" i.e. hollow sections surrounded by a solid skin. The cinpress process can be used on mold designed for structural foam molding although the best results are achieved in molds specially designed for the process.

As with structural foam molding, the mold is injected with a "short shot" and it is the pressure of the gas which forces the plastic against the mold and thus there are no sink marks. However, cycles times are reported to be only about half of those on similar structural foam moldings. But in case of saving material as well as cost of the process another method of molding is very common known as sandwich molding.

5.8 Sandwich Molding

This is an injection molding method which permits material costs to be reduced in large moldings. In most molding it is the outer surface of an article which is important in term of performance in service. If an article has to be thick in order that it will have an adequate flexural stiffness then the material within the core of the article is wasted because its only function is to keep the outer surface apart. The philosophy of the sandwich molding is that two different materials should be used for the core and the skin. That is, an expensive high performance material is used for the skin and a low-cost commodity or recycled plastic is used for the core. Initially the skin material is injected but not sufficient to fill the mold. The core material is then injected and it flows laminarly into the interior of the core. Finally the nozzle valve rotates so that the skin material is injected into the core thereby clearing the valve of the core material in preparation for the next shot . In a number of the cases the core material is foamed to produce a sandwich section with a thin solid skin and a cellular core. It is interesting that in the recent applications of sandwich molding it is the core material which is being regarded

as the critical component. This is to meet design requirements for the computers, and electronic equipments.

Plastic with the high loading of conductive filler means that the surface finish is poor and unattractive. To overcome this the sandwich molding techniques can be used in that a good quality surface can be molded using a different plastic.

5.9 Thermoforming

When the thermoplastic sheet is heated it becomes soft and pliable and the techniques for shaping this sheet are known as thermoforming. In the early stages of this method it is only limited to packaging applications. In recent years, however, there have been major advances in machine design and material availability with the result that although packaging is still the major market sector for the process, a wide range of the other products are made by thermoforming. These include aircraft window reveals, refrigerator liners, baths, switch panels, car bumpers, motorbike fairings.

The term thermoforming incorporates a wide range of possibilities for sheet forming but basically there are two sub-divisions, vacuum forming and pressure forming.

5.9.1 Vacuum Forming

In this processing method a sheet of thermoplastic material is heated and then shaped by reducing the air pressure between it and a mold. The simplest type of vacuum forming is known as negative forming and is capable of providing the depth of draw which is $1/3-1/2$ of the maximum width. The principle is very simple.

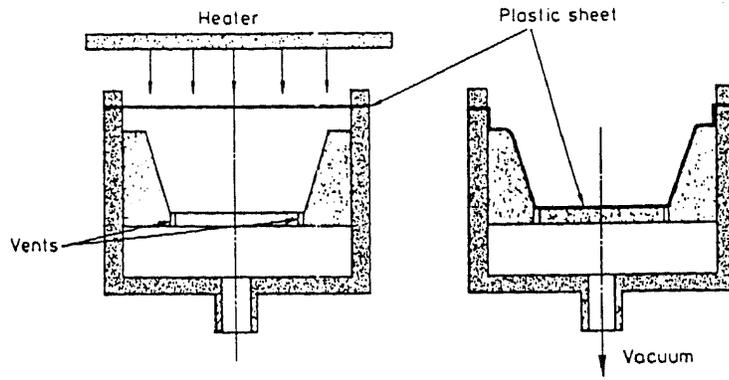


Figure 11 Vacuum Forming

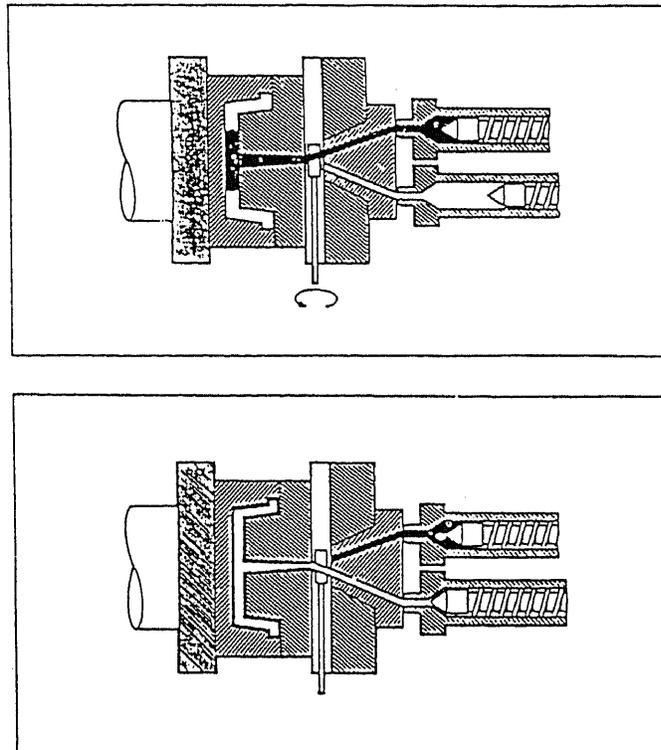


Figure 10 Stages in Liquid Injection Molding

vacuum is applied. For the thicker sheets it is essential to have heating from both sides.

In some cases negative forming would not be suitable because the shaped formed would have a wall thickness in the corners which is considerably less than that close to the clamp. If this was not acceptable then the same basic shape could be produced by positive forming. In this case the male mold is pushed into the heated sheet before the vacuum is applied. This gives the better distribution of the material and deeper shapes can be formed, depth to width ratios of 1:1 are possible. This method is also called as drape forming.

Another alternative would be to have a female mold but after the heating stage and before the vacuum is applied, a plug comes down and guides the sheet into the cavity. When the vacuum is applied the base of the molding is subjected to less draw and the result is the more uniform thickness of the wall. This is called plug assisted forming. In the packaging industry skin and the blister vacuum machines are used. Blister packs are performed foils which are sealed to a rigid backing card when the goods have been inserted. The heaters used in thermoforming are usually of the infra red type with the typical loadings of between 10 and 30 kW/m. Normally extra heat is concentrated at the clamped edges of the sheet to compensate for the additional heat losses in this region. The key to successful vacuum forming is achieving uniform heating over the sheet. One of the major attraction of the vacuum forming is that since only atmospheric pressure is used to do the shaping, the mold do not have to be very strong. Materials such as plaster, wood and thermosetting resins have all been used successfully. However, in long production runs mold cooling becomes essential in which case a metal mold is necessary.

Materials which can be vacuum foamed satisfactorily include polystyrene, ABS, PVC, acrylic, polycarbonate, polypropylene and high and low density

polyethylene. Co-extruded sheets of the different plastics and the multi-color laminations are also widely used nowadays. One of the most recent developments is the thermoforming of crystallisable PET for, the high temperature applications such as oven trays. The PET sheet is manufactured in the amorphous form and then during thermoforming it is permitted to crystallise. The resulting molding is thus capable of remaining stiff at elevated temperatures.

CHAPTER 6

FEED SYSTEM

6.1 Introduction

It is necessary to provide a flow-way in the injection mold to connect the nozzle (of the injection machine) to each impression. This flow way termed the feed system. Normally the feed system consists of a sprue, runner and gate. These terms apply equally to the flow-way itself, and to the molded material which is removed from the flow-way in the process of extracting the molding.

A typical feed system for a four-impression, two plate-type mold . It is seen that the material passes through the sprue, main runner, branch runners and gate before entering the impression. It is desirable to keep the distance that the material has to travel down to the minimum to reduce pressure and heat losses. It is for this reason that careful consideration must be given to the impression layout.

The purpose of the cold slug well, shown opposite the sprue, is theoretically to receive the material that has chilled at the front of the nozzle during the cooling and the ejection phase. Perhaps of the greater importance is the fact that it provides positive means whereby the sprue can be pulled from the sprue bush for ejection purposes. The most important factor in this system is runner, entire system is heavily depends upon the designing of runner.

6.2 Runner

The runner is a channel machined into the mold plate to connect the sprue with the entrance, to the impression. In the basic two plate mold the runner is positioned on the parting surface while on the more complex designs the runner may be positioned below the parting surface.

The wall of the runner channel must be smooth to prevent any restriction to flow. Also, as the runner has to be removed with the molding, there must be no marks left which would tend to remain the runner in the mold plate. To ensure that these points are met, it is desirable for the mold designer to specify the runner is polished in line of draw.

There are some other considerations for the designer to bear in mind:

- (1) The shape of the cross-section of the runner
- (2) The size of the runner
- (3) The runner layout

6.2.1 Runner Cross-Section Shape

The cross-sectional shape of the runner used in a mold is usually one of the four forms:

1. Trapezoidal
2. Fully round
3. Modified trapezoidal
4. Hexagonal

The criterion of efficient runner design is that the runner should provide a maximum cross-section area from the standpoint of the pressure transfer and a minimum contact on the periphery from the standpoint of the heat transfer. The ratio of the cross-sectional area to periphery will, therefore, give a direct indication of the efficiency of the runner design, the higher the value the greater the efficiency.

Ratios for various types of the runner are given in the manual. The round and square types of the runner are the two most satisfactory designs from this standpoint, whereas the ratios exhibited by the semicircular and rectangular

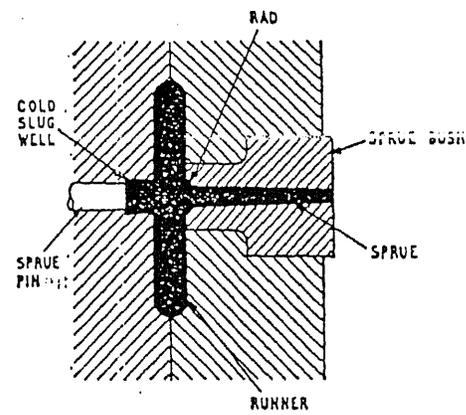
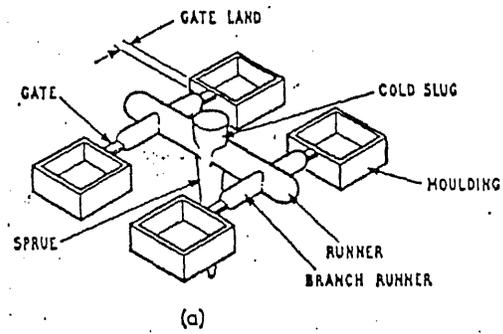


Figure 12 Feed System

types make their use generally undesirable. Unfortunately the square runner is not satisfactory for the other reason, it is difficult to eject. In practice, because of this, an angle of 10 degrees is incorporated on the runner wall, thus modifying the square to the trapezoidal section. The volume of the trapezoidal runner is approximately 25% greater than that of a round runner with the corresponding dimensions. To reduce this difference and still maintain corresponding dimensions, a modified trapezoidal form has been developed in which the volume is only 14% greater than its round counterpart.

The hexagonal runner is basically a double trapezoidal runner, where the two halves of the trapezium meet. Naturally if the similar cross-sectional areas are required, then the value of the diameter must be increased according to condition. Some tool makers feel that it is easier to match the two halves of the hexagonal runner compared with matching the two halves of a round runner. This point applies particularly to the runners which are less than 3 mm (1/8 in) in width.

As the plastic melt progresses through the runner and the mold system the melt adjacent to the cold mold surface will rapidly decrease in temperature and solidify. The material which follows will pass through the center of this solidified material and, because of the low thermal conductivity that most thermoplastics possess, the solidified material acts as an insulation and maintains the temperature of the central melt flow region. Ideally, the gate should therefore be positioned in line with the center of the runner to receive with the fully round runner and also with the hexagonal runner.

The main objection to the fully round runner is that this runner is formed from two semicircular channels machined one in each of the mold plates. It is essential that these channels are accurately matched to prevent an undesirable and

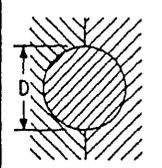
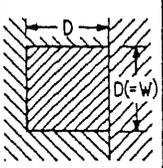
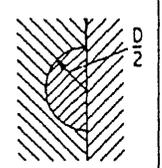
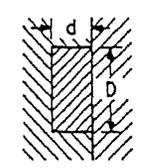
												
ROUND	SQUARE	SEMICIRCULAR	RECTANGULAR									
0.25D	0.25D	0.153D	<table border="1"> <tbody> <tr> <td>d =</td> <td>$\frac{D}{2}$</td> <td>0.166D</td> </tr> <tr> <td></td> <td>$\frac{D}{4}$</td> <td>0.1D</td> </tr> <tr> <td></td> <td>$\frac{D}{6}$</td> <td>0.071D</td> </tr> </tbody> </table>	d =	$\frac{D}{2}$	0.166D		$\frac{D}{4}$	0.1D		$\frac{D}{6}$	0.071D
d =	$\frac{D}{2}$	0.166D										
	$\frac{D}{4}$	0.1D										
	$\frac{D}{6}$	0.071D										
AREA PERIPHERY RATIO												

Figure 13 Efficiency of Various Runner Profiles

inefficient runner system being developed. A similar argument applies to the hexagonal runner system. The fact that these channels must be accurately matched means that the mold cost for a mold containing round or hexagonal runners will be greater than for one containing trapezoidal runners.

The choice of the runner section is also influenced by the question whether positive ejection of the runner system is possible. The case of a two plate mold in which a circular runner has been machined from both parting surfaces. In this case, as the mold opens, the runner is pulled from its channel in one mold half and it is then ejected from the other mold half either directly, by ejector pin, or by relying on its attachment to the molding by the gates.

For multi-plate molds, however, positive ejection of the runner system is not practicable. Here the basic trapezoidal-type runner is always specified, the runner channel being machined into the injection half from which it is pulled as the mold opens. In this way the runner is free to fall under gravity between mold plates. If the circular runner had been specified, the runner system could well adhere to its channel and make its removal difficult. Sum up the points concerning cross-sectional shape, we can say that for a simple two-plate mold which has a flat parting surface the fully round or hexagonal runner is to be preferred, the increase mold cost being relatively small. For molds which have complex parting surfaces, where it would be difficult to match accurately the semicircular channels of the round runner or, for multi-plate molds, the trapezoidal or modified trapezoidal section should be used.

6.3 Runner Size

When deciding the size of the runner the designer must consider the following factors:

1. The wall section and the volume of the molding

2. The distance of the impression from the main runner or sprue
 3. The runner cooling considerations
 4. The range of the mold maker's cutters available
 5. The plastic material to be used.
1. The cross-sectional area of the runner must be sufficient to permit the melt to pass through and fill the impression before the runner freezes and for the packing pressure to be applied for shrinkage compensation if required. Because of this, runners below 2 mm ($3/32$ in) diameter are seldom used and even this diameter is normally limited to branch runners under 25 mm (1 in) in length.
 2. The further the plastic melt has to run along the runner the greater is the resistance to flow. Hence the distance the impression is from the sprue has a direct bearing on the choice of the cross-sectional size of the runner.
 3. The cross-sectional area of the runner should not be such that it controls the injection cycle, although this is sometimes unavoidable for very light moldings. The larger the cross-sectional area of the runner the greater is the bulk of the material it contains and the longer the period it takes to cool sufficiently to enable the mold to be opened and the moldings and runner ejected. For this reason it is desirable to make the runner larger than 10 mm ($3/8$ in) diameter for most materials.
 4. The size chosen for the runner should be in a range consistent with the moldmaker's not having to carry in stock a multitude of different sizes of cutters. In practice the following are the most common sizes: 2-13 mm in 1 mm steps in the metric range and $1/8$ - $1/2$ in with $1/16$ in steps in the imperial unit range.

The plot of diameter versus length of the runner for various weight of the molding, adopting the metric system of dimensioning. The corresponding plot using the imperial dimensioning system.

Theoretically the cross-sectional area of the runner should be equal to, or in excess of, the combined cross-sectional areas of the branch runners that it is feeding. This relationship is not valid if maximum suggested diameter is reached.

6.4 Runner Layout

The layout of the runner system will depend upon the following factors:

1. The number of the impressions
2. The shape of the components
3. The type of the mold
4. The type of gate

The runner length should always be kept to a minimum to reduce pressure losses, and the runner system should be balanced. Runner balancing means that the distance the plastic material travels from the sprue to the gate should be the same for each molding. This system ensure that all the impressions fill uniformly and without interruption providing the gate lands and the gate areas are identical. It is not always practicable that to have a balanced runner system and this particularly applies to molds which incorporate a large number of differently shaped impressions. In these cases balance filling of the impression can be achieved by varying the gate dimension, that is by balanced gating.

6.4.1 Single Impression Mold

Single impression molds are usually fed by a direct sprue feed into the impression and hence no runner system is required. It may be desirable to edge gate in which case a short runner may be used. But note that by gating a single impression in this way the impression itself must be offset. This is undesirable, particularly with the large impression, as the injection pressure will exert an

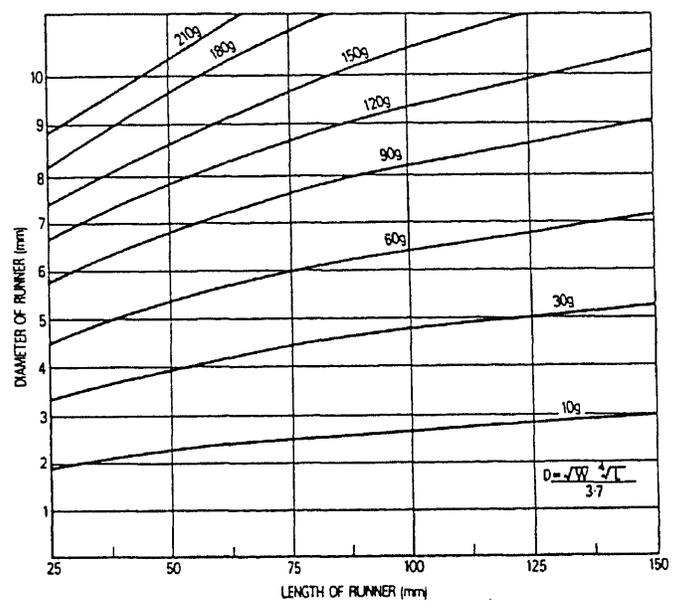


Figure 14 Guide to Runner Diameter for Molding Section

unbalanced force which will tends to open the mold one side and may result in flashed molding.

6.4.2 Two Impressions Mold

The various alternatives for feeding two impression are shown in figure 16. The simplest case is where the runner takes the shortest path between the two impressions. But it is not always possible to adopt this short runner system. Because the most desirable position of the gate may not be the center-line of the mold. If we consider the plan of a mold for two rectangular blocks, it is seen that solely from the viewpoint of mold layout it is desirable to have a impressions positioned as shown with short runners to the sides of the impression, thus enabling the size of the mold to be kept to a minimum.

However, there are other considerations, such as that of correct gating, and it may be desirable to gate at one end of the impression. To achieve these end gates it is necessary to alter the design of the layout, so that either T-shaped runner extend beyond the impressions and is then connected to the gates by short branch runners, or the runners in the form of an S, sweeps round to gate , without the necessity for the branch runners. In general, providing that the impressions are approximately the same size and shape, no difficulty should be experienced in designing balanced runner systems in case of two impression molds, but sometimes it is very difficult to apply this system, therefore other impression mold should be designed to overcome all sort of problems.

6.4.3 Three Impression Mold

A balanced runner system for the three similar impressions. In this case the impressions are placed on a pitch diameter 120 is essential for the requirement

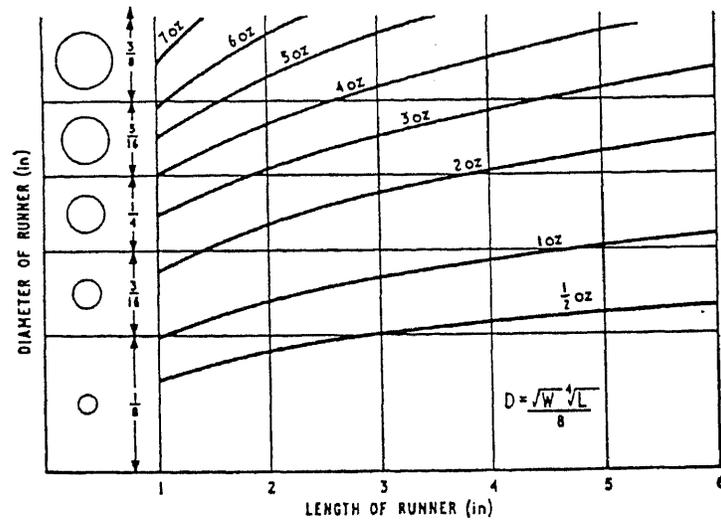


Figure 15 Guide to Runner Diameter for Molding Sections

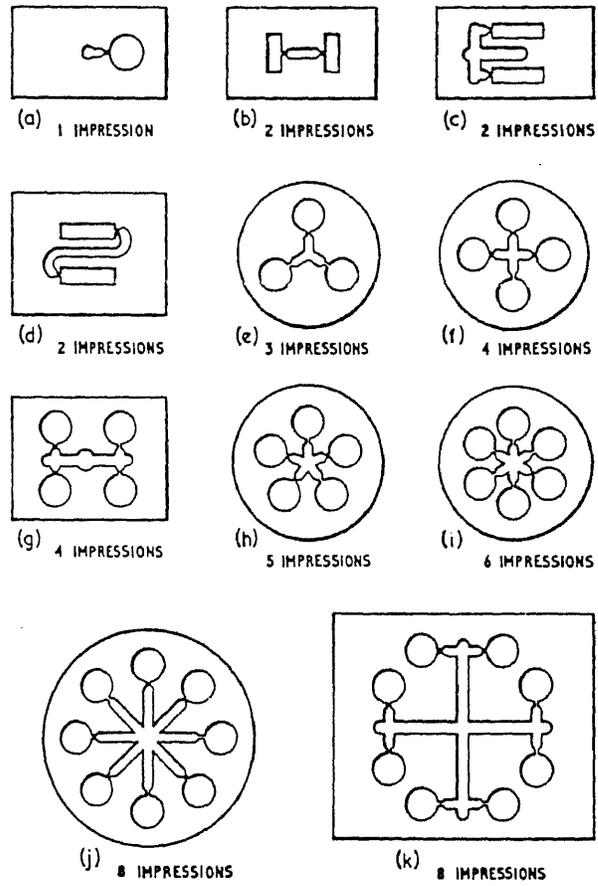


Figure 16 Balanced Runner Layouts

degrees apart, this design allows the runners to be kept to a minimum length. But in the case of different shapes and sizes of the impressions it is very difficult to balance the system without having a different system. For such cases, the large impression is being fed directly from the sprue via a short runner, while the smaller components are fed via a branch and main runner system.

6.5 Other Multi-Impression Molds

For four or more impression molds the design of the runner layout is simply an extension of the previous discussion.

As the number of the impression increase, however, the pitch circle diameter design becomes progressively more impracticable as the runner length, which is a function of the pitch circle diameter, is also increased. This results in large diameter runners being required which progressively lengthens the molding cycle and more scrap is produced. When the large number of impressions have to be accommodated, or where the impressions are of the greatly dissimilar shape, the alternative main and the branch runner system is usually adopted.

CHAPTER 7

DESIGN OF THE GATES

7.1 Introduction

The gate is a channel or orifice connecting the runner with the impression. It has the small cross-sectional area when compared with the rest of the feed system.

The presence of this gate is necessary because of several reasons:

1. The gate freezes soon after the impression is filled so that the injection plunger can be withdrawn without the probability of void being created in the molding by suck-back.
2. It allows for simple degating and in some molds this degating can be automatic.
3. After degating only a small witness mark remains.
4. Better control of the multi-impressions can be achieved.
5. Packing the impression with material in excess of that required to compensate for shrinkage is minimized.

The size of the gate can be determined in terms of the gate cross-sectional area and the gate length, the latter being known as the gate land. The optimum size of the gate depends on a number of factors.

1. The flow characteristics of the material to be molded.
2. The wall section of the molding.
3. The volume of the material to be injected into the impression.
4. The temperature of the melt
5. The temperature of the mold.

Normally the gate size is chosen on the basis of past experience, but there are few factors which may decide the dimensions of a required gate.

7.2 Location of the Gate

The location of the gate is very important and it should be such that there is an even flow of the melt in the impression, so that it fills the impression uniformly and the advancing melt front spreads out and reaches the various impressions extremities at the same time. In this way two or more advancing fronts would rarely meet to form a weld line with consequent mechanical weakness and surface blemish in the molding.

Such an ideal position for the gate is possible in some cases such as those with circular cross-sections, for example, a cup or a cone in which material is fed through the center of the base or apex. Another reason for central gating at the apex of the slender cone-like components such as pen caps is that side gating may cause deflection of the core. This can arise because side gating gives rise to a more rapid flow of the material down one side of the impression, resulting in the differential pressure which can move the core out of the position. This results in a thinner wall section on one side, thus adding another weakness to that of the weld line. The position of the melt front in the partly filled impression when the alternative gate positions are used.

For rectangular moldings the ideal position does not exist but the central gating is considered the best. But in the case of thin-walled rectangular moldings, particularly when the material used can be exhibited differential shrinkage causing distortion, off-center multi-point gating or film gating is advantageous.

When the edge gate is used, and the majority of moldings are edge gated for reason of mold economics, the gate should be positioned so that the melt flow immediately meets a restriction. An example of incorrect gating of a solid rectangular block type of molding. The impression is fed in the center at one end and the material, on entering at high velocity, 'jets' and quickly sets reaching the cool mold walls. More material then enters and flow

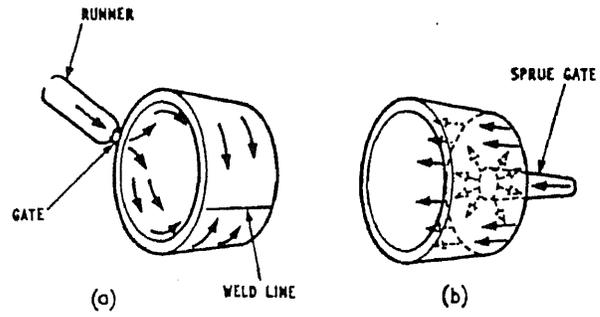


Figure 17 Position of Gate

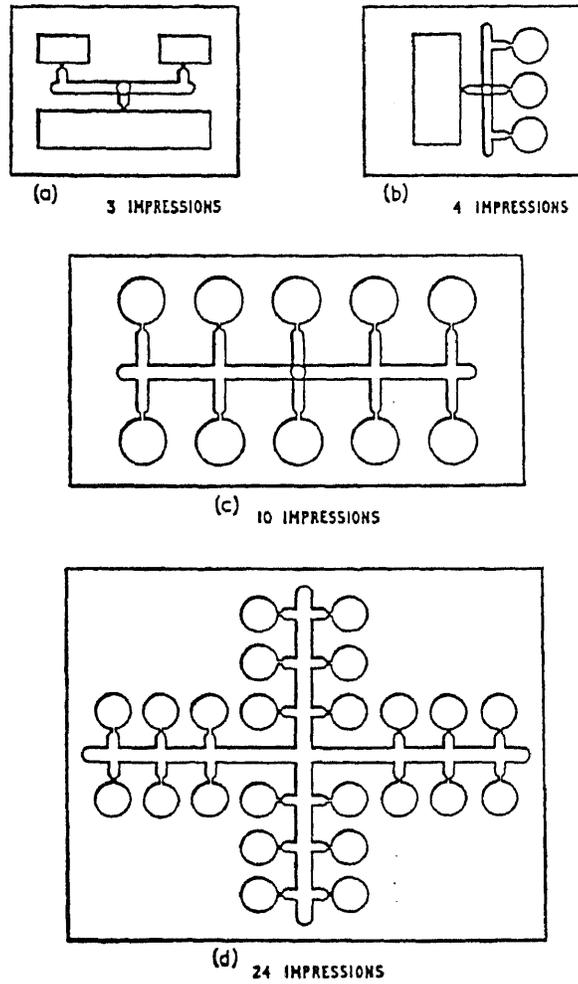


Figure 18 Runners Layouts Which Necessitate Balanced Gating

around the original jetted material. The resulting flow lines are often visible on finished molding. The trouble in this particular case can be overcome by overlap feeding. The flow of the material issuing from the gate is forced to impinge on an opposing face of the impression and this causes the material to form an advancing front which progressively fills the impression displacing the air in front of it, thus forming a molding free of flow lines.

The box type of component, which necessitates a cavity and core, automatically provides opposition to jetting so that the edge gate here is quite permissible. However, some weld lines are to be expected where the two flows meet on the opposite side of the impression. In such cases, although flow lines cannot be prevented, their effects, such as mechanical weakness and surface blemish, can be largely overcome. This is accomplished by keeping the cooling medium away from the neighborhood of weld lines, the mold temperature then increases slightly at these points, helping the two fronts to knit together more easily.

7.3 Balanced Gating

The balance of the gate for multi-impression mold is necessary so that it will ensure the filling of the impressions in a balanced manner, that is the impressions fill simultaneously. This method is adopted when the preferred balanced runner system cannot be used. Consider the runner system. The melt will take the easiest path, hence once the runner system is filled, those impressions closest to the sprue will tend to fill first and those at the greatest distance to fill last. Sometimes impressions may be overpacked while others may be starved of material. To achieve balanced filling of these impressions it is necessary to cause the greater restrictions to the flow of the melt to those impressions closest to the sprue and to progressively reduce the restriction as the distance from the sprue

increases. By adopting the method of balancing gating there are two ways of varying the restriction.

1. By varying the land length
2. By varying the cross-sectional area of the gate.

In practice balanced gating is the matter of trial and error, the land length is normally kept constant, starting with the small gate width, the mold is tried out with the short injection stroke so that short moldings are obtained. On inspection it will be obvious which impression are filling first. The gate width can then be progressively enlarged and adjusted until balanced filling is achieved.

7.4 Types of Gates

For the achievement of the best filling conditions the type of the gate is most important factor which should be chosen carefully.

7.4.1 Sprue Gate

When the molding is directly fed from a sprue or secondary sprue, the feed section is termed a sprue gate. The main disadvantage of this type of gate is that it leaves a large gate mark on the molding. The size of this mark depends on :

1. The diameter at the small end of the sprue.
2. The sprue angle.
3. The sprue length.

Thus the gate mark can be minimized by keeping the dimensions of the above factors to be minimum. Note that the as the sprue entry is controlled by the nozzle exit diameter, and as it is undesirable to reduce the sprue angle below two degrees inclusive for withdrawing purposes, the sprue length is the logical dimension for the designer to attempt to reduce. An extension nozzle can often be used to advantage, because it enters a recess in the mold and cuts down the

overall sprue length. One basic two-plate molds the sprue gate is used only for single-impression molds. In this case the impression is positioned in the center of the mold and the sprue is a direct feed into it.

A modified form of sprue gate is also used on underfeed molds and runnerless molds. In both the cases, any number of impressions can be accommodated and the sprue gates, now termed secondary sprues, are fed from runner systems situated below the parting surface.

7.4.2 Rectangular Edge Gate

This a general purpose gate and its simplest form is merely a rectangular channel machined in one mold plate to connect the runner to the impression. A section through the relevant parts of a typical mold.

This gate offers certain advantages over many other form of the gates:

1. The cross-sectional form is simple and, the fore cheap to machined.
2. Close accuracy in the gate dimensions can be achieved.
3. The gate dimension can be easily and quickly modified.
4. The filling rate of the impression can be controlled relatively independently of the gate seal time.
5. All common molding materials can be molded through this type of the gate.

One disadvantage of this type of the gate , is that after gate removal a witness mark is left on a visible surface of the molding. This is more noticeable with certain materials, particularly if the moldings are simply broken from the gates. To reduce the possibility of the gate shatter marks on the molding the modified form to gate shown at (d) may be used. For certain products, such as self assembly toy kits, a projecting gate is best avoided. In these case adopt the alternate modified design (e).

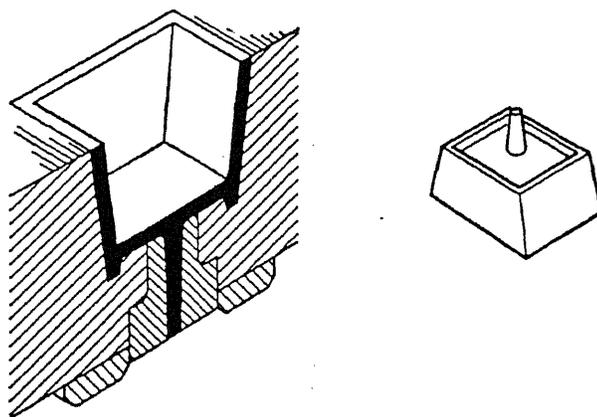


Figure 19 Sprue Gate

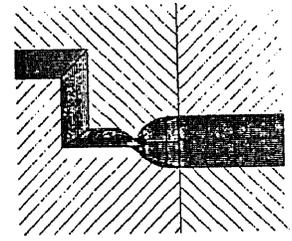
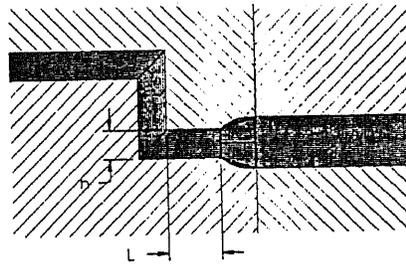
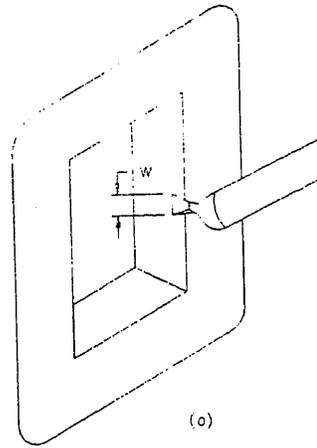


Figure 20 Different Shapes of Gate

7.4.3 Gate Size

Because of the rectangular shape, the dimensions of the gate are given by width (W), depth (h), and length (L). Now the pressure drop across the gate is approximately proportional to the length and therefore this should be kept as small as possible consistent with the strength of the steel which remains between the runner and the impression. In practice a value of between 0.5 mm (0.02 in) and 0.75 mm (0.03 in) is satisfactory.

The minimum depth of the gate controls the time for which the gate remains open. This gate open time must be sufficient for the material to reach the extremities of the impression. Now providing that the wall section of the component has been correctly chosen with respect to the maximum length of flow required, it appears reasonable to expect a relationship between the gate depth dimension and the wall section of the component.

In practice the following empirical relationship for gate depth has been found useful:

$$h = nt$$

where h = depth of gate mm (in)

t = wall section thickness mm (in)

n = material constant

While theoretical it is probable that each material should have a different value for "n", in practice it is convenient to group certain materials together and to use a group constant.

Group 1: polythene, polystyrene

Group 2: polyacetal, polycarbonate, polypropylene

Group 3: cellulose acetate, polymethyl methacrylate, nylon

Group 4: PVC

For group 1, the value of $n = 0.6$

For group 2, the value of $n = 0.7$

For group 3, the value of $n = 0.8$

For group 4, the value of $n = 0.9$

The cross-sectional area of the gate ($h \cdot W$) controls the rate at which the plastic material enters the impression. If the concept of the gate depth, wall-section relationship is accepted then the gate depth is established first. This means that the width of the gate becomes the controlling dimension of the flow rate.

The gate width is therefore usually based upon experience gained when molding components of similar shape and size.

The empirical relationship for this width is given by:

$$W = (n/30) A/2$$

where W = gate width (mm or in)

A = surface area of cavity (mm.mm or in.in)

n = material constant.

7.4.4 Overlap Gate

This gate can be considered as a variation of the basic rectangular type gate and is used to feed certain type of molding. From the previous knowledge of this type, we have noted that the melt jets into an impression if it does not contact a restriction immediately. Therefore for block type moldings the rectangular gate is replaced by the overlap gate which, by virtue of its position, direct the melt flow against an opposite impression face. The overlap gate, which is of a general rectangular shape, is machined into the plain mold plate, in such a way that it bridges the gap between the end of the runner and the end wall of the impression.

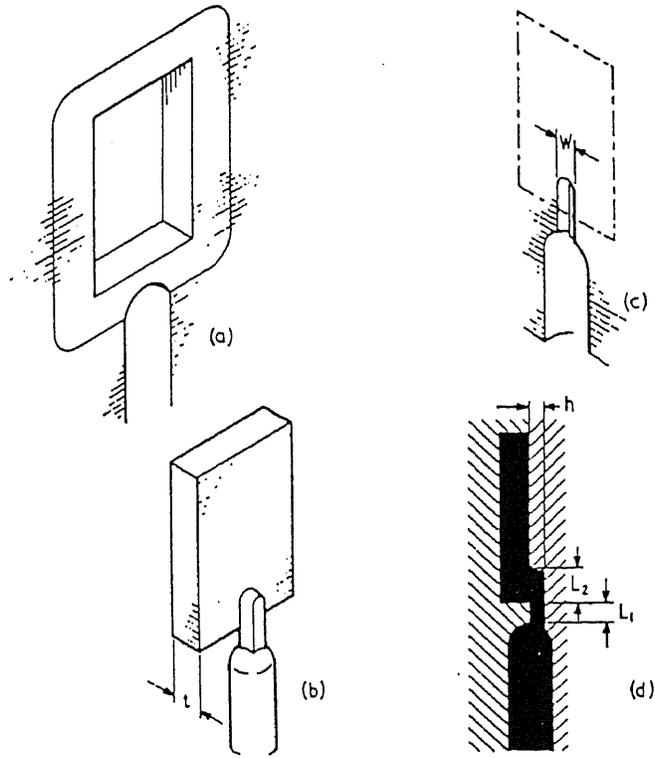


Figure 21 Different Views of Overlap Gate

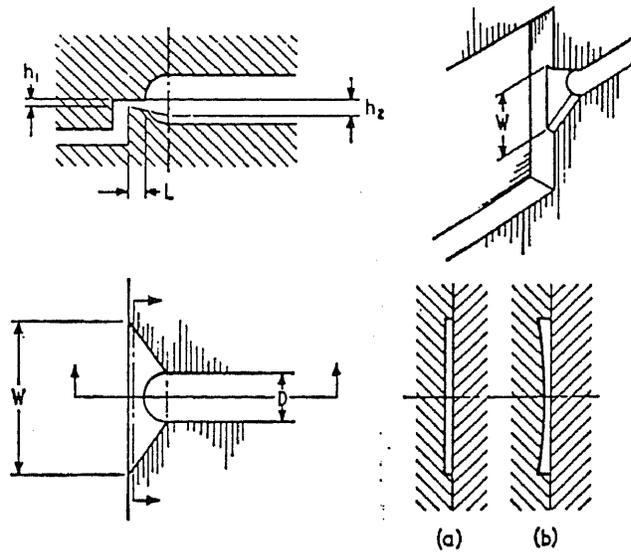


Figure 22 Different Views of Fan Gate

The gate may be used for all the common molding materials apart from rigid PVC. The gate being attached to the molding surface, does require more careful removal and finishing than for a edge gates. The size of the gate can follow the general pattern suggested for the rectangular gate with the same limitations.

Land length (distance between the end of the runner and the wall of the impression)

$$L1 = 0.5-0.75 \text{ mm}$$

gate width:

$$W = (n/30) A/2$$

gate height:

$$h = nt$$

gate length:

$$L2 = h + W/2$$

7.4.5 Fan Gate

Unlike the rectangular gate which has the constant width and depth, the corresponding dimensions of the fan gate are not constant. The width increases while the depth decreases so as to maintain a constant cross-sectional area throughout the length of the gate.

The gate at the impression is relatively wide and , because of this, a large volume of the material can be injected in a short time. This type of the gate can therefore be used advantageously for large area, thin walled moldings. The fan shape appears to spread the flow of the melt as it enters the impression and a more uniform filling is obtained with fewer flow lines and surface blemishes. This gate may be used with all the conventional molding materials apart from the certain grades of rigid PVC. The relevant gate sizes which must be decided upon are the land length, the gate width and the gate depth.

The land length needs to be slightly longer than for the rectangular gate and a suggested size for this is 1.3 mm (0.050 in). The main disadvantage of this type is that a large witness mark is left on the molding which must subsequently be trimmed and finished. It is therefore advantageous to design the gate relatively narrow and widen it only if necessary.

The height of the gate can be determined by this formula:

$$h = nt$$

To maintain a constant cross-sectional area, because of the gate form, the depth of the gate must be progressively increased back to the runner. The depth of gate at this point is given by.

$$h_2 = Wh_1/D$$

The effective length of the gate land between the runner and the impression progressively increases from a minimum at the center line to a maximum at the outer gate wall. To compensate for the increase the depth of the gate at either side to provide for more even flow through the gate.

7.4.6 Tab Gate

This is a particular gating technique for feeding solid block type moldings. A projection or tab is molded on to the side of the component and a conventional rectangular edge gate feeds this tab. The sharp right-angled turn which the melt must take prevents the undesirable 'jetting' which would otherwise occur. The melt is thereby caused to advance in a smooth steady flow and, providing the shape of the impression allows it, the impression will fill uniformly. Thus the tab gate is the alternative to the overlap type gate. The choice of one gate design or the other will depend mainly upon whether the witness mark left by the gate is best, from the appearance point of the view, on the top or on the side. Both gates leave relatively large mark of witness. This gate is used particularly for the

acrylics, but maybe used for any of the common molding materials. The size of the gate conveniently divides into two sections:

- (1) The size of the rectangular gate and
- (2) The size of the tab.

For all practical purposes the rectangular gate size can be calculated using the empirical formula for fan gate.

7.4.7 Diaphragm Gate

This gate is used for single impression tubular shaped moldings on two-plate molds. It may also be used in a similar manner for multi-impression tubular shaped moldings on underfeed and runner-less molds.

The sprue leads into a circular recess, slightly smaller than the inside diameter of the tube. This recess forms disc of material and acts as a runner which allows material to flow radially from the sprue to the gate. The gate may be cut either on the core (inset(a)) or in the cavity (inset(b)). In both cases it connects the disc runner with the impression.

The choice of the gate (a) or (b) will depend upon the use of the tube is to be put to. If the internal bore is important then the gate should be cut in the cavity. Thus by the simple machining operation on the face of the molding the bore diameter is not disturbed. Alternatively, if the internal bore is not important (a) is the better choice as the gate is more easily removed by the blanking operation.

The gate dimension which must be considered are , for method (a) land length (L) and the depth of the gate (h), and for the method (b) land length (L), overlap length (L1), depth

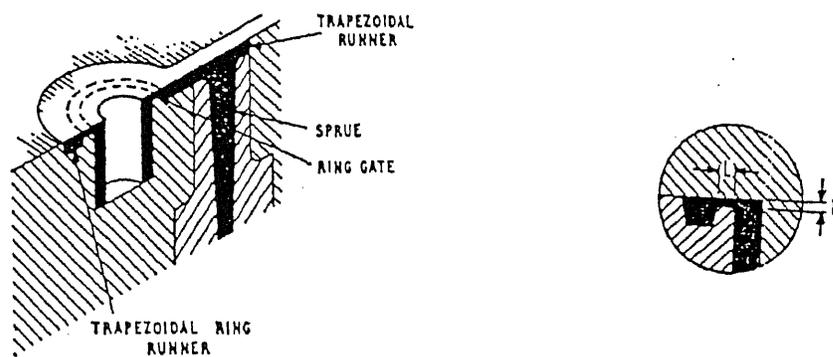


Figure 23 Different Views of Ring Gate

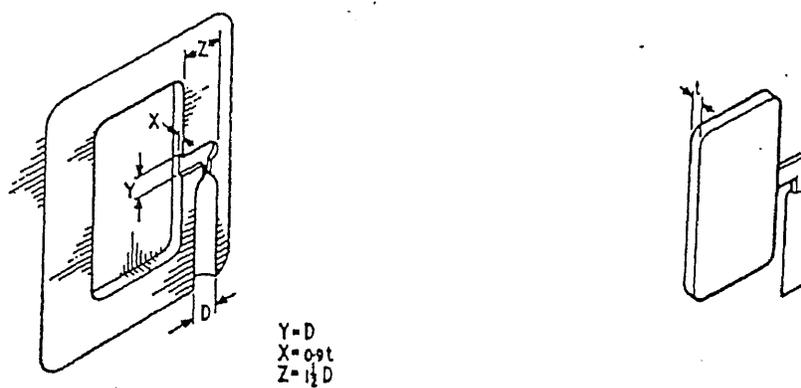


Figure 24 Different Views of Tab Gate

of the gate (h_1). Consider first the side feed diaphragm gate (a). The land length again should be a minimum consistent with the strength of the steel left between the circular runner and the impression. A value of between 0.75 mm (0.030 in) and 1 mm (0.040 in) is suggested.

The depth of the diaphragm gate is normally made slightly less than the values recommended for the rectangular gate as the corresponding values for W in this case (the inside surface of the molding) is large. The following relationship, with reference to section on the rectangular edge gate has been found to be suitable.

$$h_1 = 0.7nt$$

For the overlap type of diaphragm gate value for L can be as above. The overlap length L_1 should be at least equal to the depth of the gate (h_1) which may be computed from

$$h_1 = nt$$

The center disc is sometimes tapered from the center towards the gate to save material and to reduce the bulk for cooling reasons. This type of the gate allows for constant filling of the molding and minimizes the formation of the weld lines. It is recommended for use with all molding materials.

7.4.8 Ring Gate

The function of this gate is identical to diaphragm gate and the same comments apply. This type of the gate is used for the tubular-type moldings when more than one impression is required in a simple two-plate mold. The gate provides for a feed all around the external periphery of the molding and permits the use of the conventional runner system to connect the impressions. The runner, in the form of the trapezoidal annulus, is machined into the mold plate. The trapezoidal

runner is normally used since this type of molding would be ejected using the stripper plate. The gate is in the form of a concentric film between the runner and the impression. The dimensions of this gate are identical to those for the diaphragm gate.

7.4.9 Film Gate

This gate may be considered as a long rectangular-type edge gate and it is used for large, thin-walled components to assist in the production of the warpage free products. The gate normally extends across the complete width of the molding, although a smaller width may be used initially, which, if it proves satisfactory, will save some finishing time. The gate is similar in principle as that of diaphragm gate and ring gate in that it provides for the large flow area and results in a quick fill time. Because of this feature, the gate depth may be somewhat less than for a corresponding rectangular gate. The same relationship as given in the ring gate is suggested.

$$h = 0.7nt$$

A runner is provided parallel to the side face of the impression to feed the gate evenly with the material. This means that the very slender wall of steel exists between the runner and the impression, and to prevent this from collapsing in use, a minimum land length of 1.3 mm (0.050 in) is suggested for this gate. It is normal practice to extend the runner beyond the end of the impression, irrespective of the gate length. The gate is useful for those materials which exhibit differential shrinkage for which central feeding is impracticable.

7.4.10 Pin Gate

This is a circular gate used for feeding into the base of the components and, because it is relatively smaller in the diameter, it is often to be preferred to the

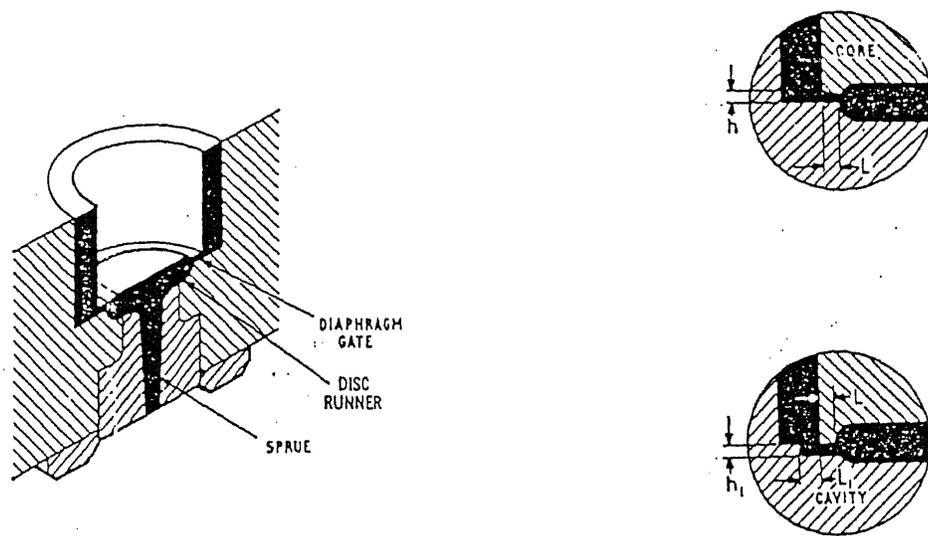


Figure 25 Different Views of Diaphragm Gate

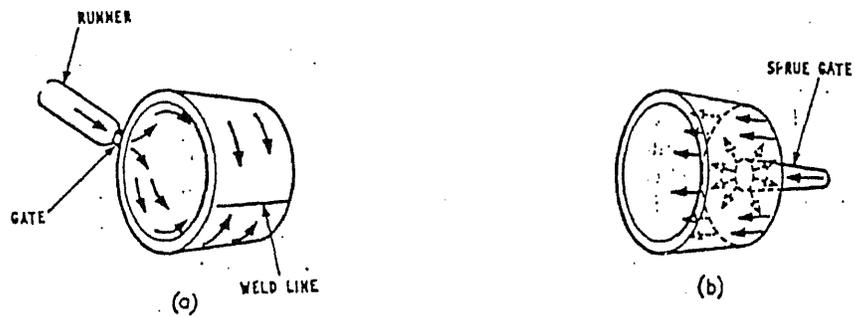


Figure 26 Different Views of Position of Gate

sprue gate which necessitates a finishing operation. However, the pin gate may only be used in certain type of molds and these are generally more complex in design than the molds in which sprue gating or side gating techniques are used. As an alternative to feeding into the center of the component. This type of the gate is often used for single, off-center feeding. This technique is particularly desirable for use with materials which exhibit differential shrinkage characteristics, and it is often used as an alternative to film gating.

To permit the use of the pin gate one of the following mold design must be adopted.

(1) Three-plate, underfeed-type mold. An extra plate is added behind the cavity plate to permit a runner system to be incorporated below the cavity or cavities. The pin gate connects the impression to the runner, either directly or via a secondary sprue.

(2) Hot-runner molds. A heated insulated runner block is incorporated behind the cavity plate. The impression is connected to the secondary nozzle, again by a pin gate and secondary sprue. Both the above designs may be used for central or offset, single or multi-point gating, into one or more impressions.

(3) Two-plate mold with special nozzles. A relatively large recess is provided behind the cavity to permit a specially designed nozzle to protrude into the mold (fig 27). The pin gate connects the impression to the nozzle usually via a hot well of plastics material. This is known as the antechamber design.

The gate dimension which must be considered are the land length (L), and the gate diameter (d). To minimize the pressure losses, as for all other gates, the land length is kept to a minimum consistent with the strength of the steel used. A land length of between 0.5 mm (0.02 in) and 0.75 mm (0.03 in) is suitable.

1. The matching form is more difficult to machine.
2. Precise dimensions are more difficult to achieve.

3. The filling rate of the impression cannot be controlled independently of the gate seal time.

Because of the above disadvantages this gate is seldom used for the molding with wall thickness below 4 mm (0.150 in). The gate is used, for thicker wall sections. For this type of molding a relatively large gate is an advantage to ensure that the gate remains open sufficiently long to allow follow-up pressure to be applied to prevent shrinkage.

7.4.11 Subsurface Gate

The subsurface gate is a circular or oval type gate which submerges and feed into the impression below the parting surface of the mold. While similar to round edge gate in that it is of similar or nearly similar shape and the feeds into the side of the impression, it has several advantages over the round gate.

1. The form, being in the one mold plate, has no matching problems and precise dimensions can be achieved.
2. If the more oval form is used the filling rate of the impression can be controlled independently of the gate seal time.
3. The gate is sheared from the molding during its ejection.

The molding and the feed systems are removed separately from the mold and this means that a separate runner ejection is advantageous, particularly as a certain amount of deformation of the runner is necessary to remove the secondary runner from the mold.

The gate dimensions which must be considered are land length which, because of the form, needs to be L (minimum) = 1.9 mm (0.075 in). Φ is the angle subtended between the centre-line of the runner and the impression wall (29). This angle is normally between 30 and 45 degrees. The dimensions of the gate cross-section can be estimated from either the equation suggested for the

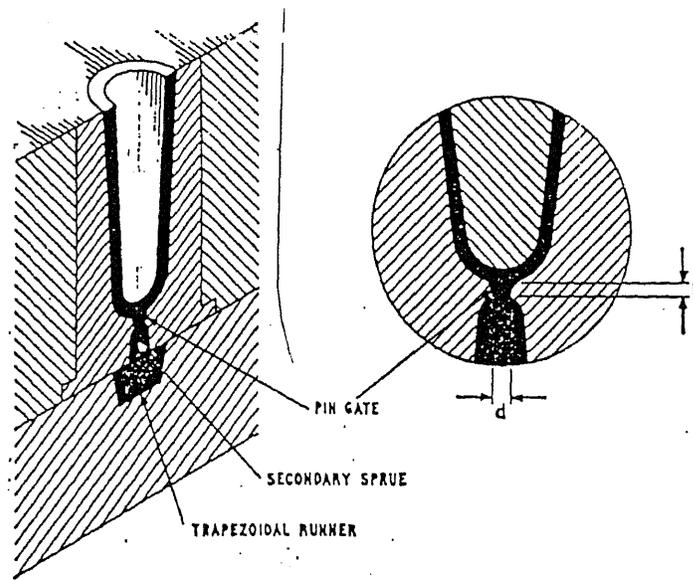


Figure 27 Different Views of Pin Gate

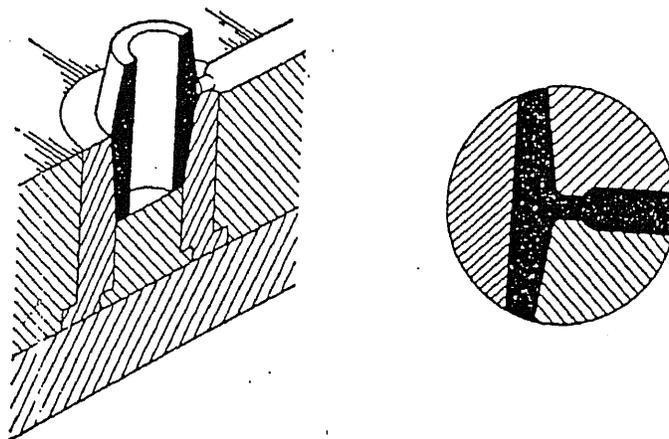


Figure 28 Different Views of Round Edge Gate

rectangular gate, if an oval gate is adopted, or the equation for the pin gate, if the circular gate is used.

The subsurface gate can be adopted for feeding into the inside surface of a component, providing at least one of the following conditions are met:

1. There is a suitable member which projects below the general parting line surface and which is also located relatively close to the component wall. The subsurface gate feeds directly into this member.
2. In design where it is possible to incorporate a small diameter peg relatively close to the component wall, the subsurface gate feeds directly into the peg and thereby into the impression. The peg is subsequently removed as a post molding operation.

CHAPTER 8

PLASTIC MOLDING MATERIALS

8.1 Introduction

Plastics may be made hard, elastic, rubbery, tough, crystal-clear, opaque, strong, stiff, outdoor-weather-resistant, electrically conductive, or practically anything that is desired, depending on the choice of the starting materials and the method of the molding. A specific plastic can be molded using different injection molding machine settings so that dimensional tolerance on a part can vary after each molding, or the machine can be set so that extremely close tolerance can be met repeatedly.

Basically certain injection molded parts can be held to extremely close tolerance of less than a thousands of an inch or down to 0.0 percent. Tolerance that can be met can go from 5 percent for 0.020 in. thick, to 1 percent for 0.500 in., to 1/2 percent for 1.000 in., to 1/4 percent for 5.000 in.

Economical production required that tolerance not be specified tighter than necessary. However after production target is to mold to as tight as possible to be more profitable. Recognize that many plastics after molded change dimension due to temperature, humidity and load. Heat treatment can significantly reduces or even eliminate these changes for certain plastics.

Dimensional accuracy that can be met depends on different factors, such as accuracy of mold and machine performance, properties of the materials, operation of the complete molding cycle, wear or damage of machine and mold, shape/size/thickness of the part, post shrinkage that can reach 3 percent of certain materials, degree of repeatability in performance of machine/mold/material.

8.2 Types of Plastics

Thermosets are generally more suitable to meet the tightest tolerance. With thermoplastics it can be more complicated. As is well known crystalline plastics (PE, PP) generally have the different rate of shrinkage in the longitudinal and traverse direction of melt flow. In turn these directional shrinkages can significantly vary due to changes injection pressure, melt heat, mold heat and the part thickness or shape. These changes can occur at different rates in the different directions. To minimize and control different tolerance consider using the highest melt heat, keep gate surrounding area where tight tolerance are required, use machine that requires at least 70 percent of the shot capacity, minimize time that melt is in the barrel that understand complete operation of the machine/mold/material to ensure part tolerance repeatability. Not every material is suitable for molded parts requiring tight tolerances.

To understand plastics, one must first appreciate and accept the polymer chemist's ability to literally rearrange the molecular structure of the plastic or polymer to provide an almost infinite variety of compositions that differ in form, appearance, properties, and characteristics.

One must also approach the subject with a completely open mind that will accept all the contradictions that make it so difficult to pin common labels on the different families of plastics, or even on the various types within a single family. In the family of polyethylene, consumers are more aware of the so called low-density polyethylene, which are flexible materials most familiar in housewares, toys, trash bags, film overwraps, and the like. But there is another type of polyethylene, called high-density polyethylene can also be produced in a flexible film form with properties quite different from low-density polyethylene film. Finally, to fully understand the plastics, one must be aware of the many different routes that the starting materials for plastics can take on the way to consumer or

industry. Here we are concerned with those resins that are supplied to the processor in the form of the granules, powder, pellets, flakes, or the liquids and are transformed by him into solid or cellular plastics products, shapes, film or sheeting, or coatings and surfaces for various substrates.

However, the same starting materials used to make these resins can take another route and end up at the textile industry, the paint industry and the adhesives industry.

Many plastics derive from the fractions of petroleum or gas that are recovered during the refining process. For example, ethylene monomer is derived, in a gaseous form, from petroleum refining gas or liquified petroleum gases or liquid hydrocarbons. Although the petroleum or gas derivatives are not the only basic source used in making feed stocks for plastics, they are among the most popular and economical in use today. Coal is another excellent source in the manufacturing of the feedstocks for plastics, and there are other materials including such unique possibilities as agricultural oils such as castor oil or tung oil derived from plant life that are also adaptable.

From these basic source come the feedstocks we call monomers. The monomers then subjected to a chemical reaction known as polymerization that causes the small molecules to link together into ever-increasing longer molecules. Chemically, the polymerization reaction has turned the monomer into polymer. The transformation from the monomer to polymer may also help you understand why the names of so many plastics material begins with the prefix "poly". Thus, a polymer may be defined as a high-molecule-weight compounds that contains comparatively simple recurring units.

Outside of the plastics field, monomers can take different routes to produce a variety of other important products, from antifreeze to fertilizers. Even within the plastic field, a single monomer can contribute to the manufacture of the variety of

different polymers, each with its own distinctive characteristics. When the styrene monomer is polymerized, it becomes a styrene polymer or polystyrene, as it is more familiarly known in its plastic form. By a more direct route, the ethylene monomer can be polymerized to produce ethylene polymer or polyethylene, another popular plastic.

Basically, there is a great deal of flexibility in the plastic manufacturing process for creating wide range of materials. The way in which the small molecules link together into larger molecules and the structural arrangement they take is one determinant of the properties of the plastics. The length of molecules in the polymer chain is a second. The type of the molecule is a third. Polymerizing two or more different monomers together is a fourth. And incorporating various chemicals or additives during or after polymerization is a fifth. Other modifying techniques are in use, and polymer chemists continue to come up with new ones. The polymer or plastic resin must next be prepared for use by the processor, who will turn it to finished product. In some instances, it is possible to use the plastic resin as it comes out of the polymerization reaction. More often, however, it goes through other steps that put it into a form that can be more easily handled by the processor and more easily run through processing equipment. The most popular solid form for plastic resin are as pellets, granules, flakes, or powder. In the hands of the processor, these solids are generally subjected to heat and pressure, melted, forced into the desired shape, then allowed to cure and set into a finished product. Plastic resins are also available as semi-solids or as liquids, for casting.

Liquids can also be used to impregnate fibrous materials, which can then be allowed to harden into so-called reinforced plastics.

Another option available to processor is to use resin incorporating a blowing agent. Subjected to the heat of processing or the heat from the chemical

reaction, these agents decompose and release gases that can turn a solid product into a foamed product.

It is very important to understand the flow criteria of melt plastic from basic feedstocks to the end-product. In most cases, the flow will proceed along these lines: feedstocks, known as monomers, are polymerized by the chemical reaction into polymers, the resin are then made into forms useful for processing, sometimes called molding or extrusion compounds, the compound now goes to the processor, who have a several techniques available to him for turning the resin into a finished product or part, into a secondary product that goes through subsequent fabricating operations, or into a coating or surfacing that can be applied to various substrates. At the processing level, plastics can also be turned into monofilaments for use in rope or household screening: or as binders, they are use with materials such as fibers or sheet of paper or sheets of wood to turn out products such as boat hulls, tabletops, or airplane wingtips.

8.3 Definition for Plastics

There is a generally accepted definition for plastics that goes likely this: any one of the large or varied group of material consisting wholly or in apart of combinations of carbon with oxygen, hydrogen, nitrogen, and other organic or inorganic elements that, while solid in the finished state, at some stage in its manufacture is made liquid, and thus capable of being formed into various shapes, most usually through the application, either singly or together, of heat and pressure.

Plastics are family of materials, not a single material, each member of which has its own distinct and special advantages. Whatever their properties and form, however, most plastics fall into one of two groups: the thermoplastics and thermosets.

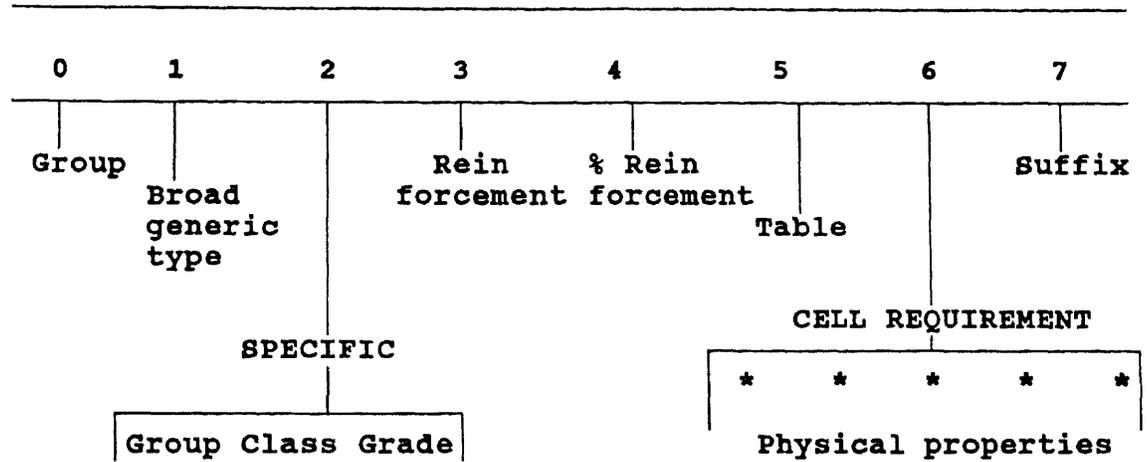
Thermoplastics resins consist of long molecules, either linear or branched, having side chains or group that are not attached to other polymer molecules. Thus, they can be repeatedly softened and hardened by heating and cooling. Usually, thermoplastic resins are purchased as pellets or granules that are softened by heat under pressure so they can be formed, then cooled, so that they hardened into the final desired shape. No chemical changes generally take place during forming. The analogy would be a block of ice that can be softened, poured into any shape of cavity, then cooled to become a solid again. In thermosetting resins, reactive portion of the molecules form cross-links between the long molecules during polymerization. The linear polymer chains are thus bonded together to form a three-dimensional network. Therefore, once polymerized or hardened, the material can not be softened by heating without degrading some linkages.

Thermosets are usually purchased as liquid monomer-polymer mixtures or as a partially polymerized molding compound. In this uncured condition, they can be formed to the finished shape with or without pressure and polymerized with chemical or heat. the analogy in this case would be to a hard-boiled egg, which has turned from a liquid to a solid and cannot be converted back to a liquid.

8.4 Identification of Plastic Materials

A classifying plastic materials standard that can serve many of the industry needs has been issued by ASTM. This standard is designated as D 4000 and entitled "Standard Guide for Identification of Plastic Material". It provides an easy means of identifying plastic materials used in the fabrication of parts. Ever since classification systems were adopted many years ago for materials such as 1030

Table 5 ASTM D 4000 Line Call-Out



- 0 = One digit for expanded group
- 1 = Two or more letters identify the generic family based on Abbreviation D 1600
- 2 = Three digits identify the specific chemical group
- 3 = One letter indicates reinforcement type
- 4 = Two digits indicate percent of reinforcement
- 5 = One letter refer to a cell table listing of physical specifications and test methods
- 6 = Five digits refer to the specific physical parameters listed in the cell table
- 7 = Suffix codes indicate special requirements based on the application, and identify special tests.

steel and elastomers, there has been an effort to issue this guide. The approach used follows the steel and elastomer unified classification systems of ASTM.

The guide provides the tabulated properties for unfilled, filled, and reinforced plastic materials suitable for processing into parts. This standard is required to reduce the growing number of material specifications, paperwork, and man-hours used to ensure that parts of known quality are being produced from commercially available materials. The D 4000 standard will eliminate the many certifications required for the same material that a processor may have to obtain from several vendors for a customer or different customers. The classification system and subsequent line call-out is intended to be a means of identifying plastic materials used in the fabrication of the end items or parts. It is not intended for the selection of materials. Material selection should be made by those having expertise in the plastic field after careful consideration of the design and performance required of the part, the environment to which it will be exposed, the fabrication process to be employed, the inherent properties of the material not covered in this document, and the economic factors.

This classification system is based on the premise that plastic materials can be arranged into broad generic families using basic properties to arrange the materials into the groups, classes, and grades.

The format to this system was prepared to permit the addition of property values for future plastics. Plastic material will be classified on the basis of their broad generic family. The generic family is identified by letter designations. These letters represent the standard abbreviations for plastics in accordance with abbreviations D 1600. The generic family is based on the broad chemical makeup of the base polymer. By its designation, certain inherent properties are specified. The generic family is classified into groups according, in general, to the

Table 6 Standard Symbols and Requirements

STANDARD SYMBOL	PLASTIC FAMILY NAME	ASTM STANDARD	UF	F
ABS	Acrylonitrile/butadiene/ styrene	D		
AMMA	Acrylonitrile/methylmetha- crylate		E	
ASA	Acrylonitrile/styrene/ acrylate		E	
CA	Cellulose acetate	D 706		
CAB	Cellulose acetate butyrate	D 707		
CAP	Cellulose acetate propionate		E	D
CE	Cellulose plastics general		E	D
CF	Cresol formaldehyde		H	H
CMC	Carboxymethyl cellulose		E	
CN	Cellulose nitrate		E	D
CP	Cellulose propionate	D 1562		
CPE	Chlorinated polyethylene		F	
CS	Casein		H	H
CTA	Cellulose triacetate		E	D
CTFE	Polymonochlorotrifluoro- ethylene			
DAP	Poly(diallyl phthalate)		H	H
EC	Ethyle cellulose		E	D
EEA	Ethylene/ethyl acrylate		F	
EMA	Ethylene/methacrylic acid		F	
EP	Epoxy, epoxide		H	H
EPD	Ethylene/propylene/diene			
EPM	Ethylene/propylene polymer		F	D
ETFE	Ethylene-tetrafluoroethy- lene copolymer			
EVA	Ethylene/vinyl acetate		F	
FEP	Perflouro (ethyle-propy- lene) copolymer			
FF	Furan formaldehyde		H	H
IPS	Impact styrene			
MF	Melamine-formaldehyde		H	H
PA	Polyamide (nylon)	D 4066		
PAI	Polyamide-imide		G	
PARA	Polyaryle amide			

Table 7 Cell Table G Detail Requirements

DESIGNATION ORDER NUMBER		PROPERTY							
1		Tensile strength, ASTM D 638 MPa, min							
		CELL LIMITS							
0	1	2	3	4	5	6	7	8	
Unspecified	15	40	65	85	110	135	160	185	
2		Flexural modulus, ASTM D 790, MPa, min							
		CELL LIMITS							
Unspecified	600	3500	6500	10000	13000	16000	19000	22000	
3		Izod impact, ASTM D 256, J/m, min							
		CELL LIMITS							
Unspecified	15	30	50	135	270	425	670	950	
4		Deflection (temperature, ASTM d 648, (1820 kPa).C, min							
		CELL LIMITS							
Unspecified	130	160	200	230	260	300	330	360	
5		To be determined							
Unspecified	

chemical composition. These groups are further subdivided into classes and grades. The letter designation applicable is followed by a three digit number indicating group, class, and grade.

The basic property tables have been developed to identify the commercially available unreinforced plastics into groups, classes, and grades. Where a standard does not exist for this classification system, the letter designated for the generic family will be followed by three o's and the use of the cell table that applies. For example, P1000 would indicate a polyimide plastic (PI), with 000 indicating no basic property, and other properties from the other table.

To facilitate the incorporation of future materials, or where the present families require expansion of the basic property table, a number preceding the symbol for the generic family is issued to indicate that additional groups have been added to the table. This digit coupled with the first digit after the generic family will indicate the group to be found in the basic property table.

Reinforced versions of the basic material are identified by a single letter that indicates the reinforcement used, and two digits that indicate the quantity in percent by mass. Thus the letter designation G for the glass reinforced and 33 for the percent of reinforcement, G33, specifies a 33 percent glass-filled material. To facilitate the identification of new, and reinforced materials where basic property tables are not provided in a material specification.

CHAPTER 9

POLYETHYLENES AND POLYPROPYLENES

9.1 Low Density Polyethylenes

Low-density polyethylenes (LDPE) is defined as polymerized ethylene having a normal density of 0.910 g/cubic cm. to 0.925 g/cubic cm. However, medium density polyethylene (MDPE), with a density of 0.926 to 0.940 g/cubic cm., is usually included with LDPE because their processing conditions and properties are quite similar. Therefore, the following information will encompass both low- and medium-density polyethylene unless otherwise stated. Specific polyethylene formulations used for illustration will be selected from the middle of the density ranges covered by LDPE and MDPE.

The injection molding of the large, thin-walled items is one of the most difficult challenges in injection molding of LDPE. At the same time, one of the most commonly encountered items of the injected molded LDPE is a lid.

9.2 Injection Molding Polyethylene Lids

Injection molded polyethylene lids are used in a wide variety of closure applications. Many products, such as margarine, cream cheese, whipped topping, ice cream, and sandwich spreads, are packed in plastic containers that have polyethylene lids for primary closure. Many other products, such as coffee, peanuts, and shortening, are packed in the metal cans that are used after they are opened to store the unused portion of the contents. Most of these cans are sold with polyethylene overcaps that snap into place and furnish good closure for the cans after removal of the metal tops. The characteristics demanded in polyethylene lids vary widely. Economy is always important, and in nearly every application, it is desirable that the lids be flat and that they cover. Some

applications demand some degree of clarity so that printed matter on a metal lid can be read through the overcap before the can is sold. Some require resistance to environmental stress cracking, so that the material that may be in contact with them will not cause them to split. Some require still other characteristics. In addition, the polyethylene lid business has undergone significant technological advancement in past years with most of the emphases on processability or production rate. Extremely fast cycling machines, stack molds, and larger tonnage presses all have contributed to an increase in the molder's productivity. Also, more sophisticated machine controls make present injection molding machines very sensitive to process and material change and/or variation.

Consequently, molding polyethylene lids for their many uses is an exacting process that requires good selection of molding machine, mold design, part design, plastic formulation, molding conditions, and other factors such as variations in processes.

9.2.1 Molding Machines

Screw type molding machines are preferred to straight-ram machines for molding polyethylene overcaps because they produce more homogeneous melts and permit the use of shorter cycles. They also permit better control of such variables as injection pressure, injection speed, and melt temperature. It is very difficult to mold flat, acceptable lid on the straight-ram machines unless it is equipped with a screw preplasticator. In the preplasticator unit, melting of the plastic is performed in a simple extruder that pumps material into a secondary cylinder containing a ram that acts as the injection unit. Such an arrangement offers excellent control of shot size because the volume of the shot is measured while the material is in the molten state.

The size of the injection molding machine to be used to mold overcaps is intimately related to the diameter of the overcap and the number of the cavities in the mold. Generally, molds with two to four cavities can be used on molding machines with capacities of two or three ounces and clamping forces of 75 to 150 tons, whereas molds with six or eight cavities frequently require machines of 5 to 16 ounces rating with 200 to 400 tons of clamping force available.

There are advantages in both large and small machines. If several small machines are used rather than fewer large ones, a machine shutdown or break for routine maintenance will have less effect on the productivity. Also, because there are normally fewer cavities in molds for small machines than in molds for large machines, smaller machines permit closer control of the molding variables in the individual cavities. On the other hand, large machines molding many lids per shot can have lower direct molding costs per part produced, even though they require longer cycles. The cycle time increases as the number of cavities in a mold increases, but not proportionately. Thus a four-cavity mold might run with a 5-second cycle, and an eight cavity mold might require an 8-second cycle, but the larger mold and the longer cycle will produce more lids per minute of operation.

The clamping force usually necessary in the molding machine for producing an overcap 25 to 30 mils thick is 1 1/2 to 2 tons per square inch of projected area. A single cavity mold for a 5-in. lid, therefore would require 28 to 30 tons, a four cavity mold for the same size lid, 110 to 150 tons. The projected area of the runner in an insulated or hot runner mold need not be considered unless its total projected area is greater than that of the lids. If this should be the case, the projected area of the runner should be considered and that of the lids ignored.

9.2.2 Molding Conditions

When condition for the molding polyethylene, the objective should be to inject fairly hot material into a cold mold while subjecting the molded part to as little strain as possible. This is usually accomplished by using higher injection pressures to ensure quick filling of the mold and by using very short plunger forward times.

9.2.3 Melt Temperature

Usually high melt temperatures are used to permit the plastics to be injected quickly into the mold with minimum strain, because high melt temperatures gives maximum clarity, minimum sunburst and minimum warpage in the molder.

On the other hand if the melt temperature is too low, molding will be very difficult, requiring extra high pressures and longer plunger-forward times. Due to this difficulty the lids produced are very poor in clarity and having sunburst and warpage.

The range for the melt temperature generally varies from 325 to 550 degrees , depending on the machine used, the mold size and the construction, and the plastic formulation. Large machines with large holdup in the cylinder usually operates between 325 to 475 degrees , whereas small machine with little hold-up generally operates between 425 and 550 degrees.

In a machine operating near its plasticating limits, an indicated temperature of 480 degrees may be required to maintain the melt temperature of 450 degrees fahrenheit.

9.2.4 Mold Temperature

The optimum mold temperature for the lid production seems to be about 40 to 50 degrees F. Temperatures in this range permit short cycles and produce lids with

good clarity. On the other hand if the temperature is below 40 degrees, it can make mold filling a difficult task. It should be noted that the very clear lids can be produced in the case of higher mold temperature results in slow lid cooling.

9.2.5 Cycle Time

The two most important factors in the lid molding cycle are plunger-forward time and clamp time.

The plunger-forward time should be about 0.1 0.3 seconds longer than the actual mold filling time. If it is significantly longer than this, the area around the gates will be packed, and thus will shrink less than the area around the outer edge of the lids, so that warpage could result. the plunger-forward time is generally determined by setting all temperatures for molding, decreasing the plunger-forward time in small increments until a short shot results, and then increasing the time about 0.1 to 0.3 second. The clamp time should be the absolute minimum setting at which the lid with acceptable flatness, toe-in, and shrinkage can be produced. The clamp time, which must be set after the plunger-forward time is fixed, must sufficiently exceed the plunger-forward time to allow the molten plastic to solidify in the cavities. Since toe-in is desirable but the warpage is not, the clamp time must be set for each mold to give the satisfaction.

9.2.6 Injection Pressure and Injection Speed

The injection pressure and the injection speed should be maximum in order to fill the cavities as rapidly as possible but keeping the proper shot-size control.

9.2.7 Shot Size

The shot size should be the exact amount of plastic needed to fill the mold cavities if it is possible, So proper adjustment of the shot size is required. On

some small molding machines, and even on some larger ones equipped with screw preplasticators, shot-size control is sufficiently precise to make this possible. With precise shot-size control, the adjustment described for cycle time, injection pressure, and injection speed should be satisfactory.

On many large machine, it is very difficult to maintain the high precision of shot-size control. One shot might be short and the other one packed without changing machine settings. If necessary, keep a cushion of the molten plastic in the injection machine to better control the rim action, but the cushion should be as small as possible.

Sometimes a cushion of molten plastic in the machine may cause excessive packing of the cavities, resulting in the warped lids. If this situation arises then it will be necessary to depart from the previously described adjustments of cycle time, injection pressure and the injection speed. The procedure is to reduce the injection pressure first, then if necessary, to reduce the injection speed. Every effort should be made to keep the rate of the molten plastic into the mold as high as possible so that no appreciable solidification of the material will occur until after the mold is filled.

9.2.8 Screw Speed

In order to get rid of the delay of cycle, maximum screw speed is usually used. Fast screw speeds generate frictional heat in the plastic and help to produce the homogeneous melt. If temperature becomes too high and the material degradation results, the screw speed should be reduced. The heat generated in the plastic by the rotation of the screw is a function of the square of the screw speed, therefore a small reduction in the screw speed can result in an appreciable reduction in the heat generated. If screw speed tends towards high side then it is very dangerous for design.

9.2.9 Materials

Tenite polyethylene 18BOA is most widely used for the production of lids. This material is characterized by excellent processability, warpage resistance, and clarity, while exhibiting good toe-in characteristics and stress crack resistance. A higher melt-index version of 18BOA is 18DOA. This material exhibits greater shrinkage and slightly better flow characteristics but does not exhibit the toe-in, processability, and the stress crack resistance of 18BOA.

Tenite polyethylene 187OA exhibits exceptional stress crack resistance. It provides the material with fast cycling characteristics for the lid molder interested in applications requiring high stress-crack resistance.

All three materials have been used extensively in the lid molding industry in a variety of closure applications and other related items.

9.3 Polypropylenes

Polypropylene and propylene copolymers are thermoplastic materials having the following characteristics:

Light weight

Ability to form an integral hinge

1. Heat resistance
2. Hardness
3. Processability
4. Surface gloss
5. Chemical resistance
6. Stain resistance
7. Stress-crack resistance

Table 9 Tenite Polyethylene 1870 Physical Properties

PROPERTY	UNIT	ASTM TEST METHOD	TYPICAL VALUE
Melt Index	g/10 min	D 1238	7.0
Density	g/cubic cm	D 1505	0.923
Softening point	centigrade	D 1525	94
Brittleness Temperature	centigrade Fahrenheit	D 746	<-78 <-108
Tensile strength at Yield	psi MPa	D 638 Type IV	1700 11.7
Tensile strength at Fracture	psi MPa	specimen	1300 9.0
Elongation at Fracture	%		200
Stiffness in Flexure	100000 psi MPa	D 747	0.35 241

Table 10 Tenite Polyethylene 18B0 Physical Properties

PROPERTY	UNIT	ASTM TEST METHOD	TYPICAL VALUE
Melt Index	g/10 min	D 1238	20
Density	g/cubic cm	D 1505	0.923
Softening point	centigrade	D 1525	92
Brittleness Temperature	centigrade Fahrenheit	D 746	<-40 <-40
Tensile strength at Yield	psi MPa	D 638 Type IV	1650 11.4
Tensile strength at Fracture	psi MPa	specimen	1300 9.0
Elongation at Fracture	%		100
Stiffness in Flexure	100000 psi MPa	D 747	0.35 241

Table 11 Tenite Polyethylene 18D0 Physical Properties

PROPERTY	UNIT	ASTM TEST METHOD	TYPICAL VALUE
Melt Index	g/10 min	D 1238	040
Density	g/cubic cm	D 1505	0.923
Softening point	centigrade	D 1525	91
Brittleness Temperature	centigrade Fahrenheit	D 746	-35 -31
Tensile strength at Yield	psi MPa	D 638 Type IV	1650 11.4
Tensile strength at Fracture	psi MPa	specimen	1300 9.0
Elongation at Fracture	%		90
Stiffness in Flexure	100000 psi MPa	D 747	0.35 241

The properties mentioned above makes polypropylene and propylene copolymers excellent choices for molding items such as housewares, appliance parts, automobile parts and accessories, closures, laboratory ware, hospital ware, toys, sporting goods, and other items for house and industry.

Polypropylene is typically supplied in either cube cut or cylindrical 1/8-in. pellets, the pellets shape depending upon the in-plant processing required for producing a particular formulation.

The plastic is offered in natural color and in a wide range of compounding colors custommatched to the user's requirements and accurately control for uniformity between lots. It can also be colored in the user's plant with either dry colors or color concentrates. Polypropylene is lighter than polyethylene and nonpolyolefin plastics and, therefore, produces more parts per pound than these other materials in any given mold. In addition the high stiffness and excellent processability of polypropylene permit the molding of parts with thin sections that would often be too flexible or unmoldable with other thermoplastics. Basic formulations of polypropylene are produced in flow rates ranging from less than 1 to 450 to meet a variety of processing and product performance requirements.

9.3.1 Physical Characteristics

In addition to low density and high stiffness, polypropylene has a high softening point and excellent chemical resistance, stress crack resistance, electrical properties, and a wide range of flow rates have promoted its use in a great variety of injection-molding applications.

9.3.2 Injection Molding Machines

Polypropylene and copolymers are well adapted to molding in any of the commercially available molding machines. These machines are screw-ram and

plunger-type machines. These machines differ in the manner in which the plastic pellets are delivered from the feed hopper to the nozzle of the machine. The effect that the screw-ram machines have on the plastic are different from those of the plunger-type machines.

Cylinder temperatures, injection pressures, and clamp pressures required for successful molding are normally lower for a screw-ram machine than for a plunger machine because the action of the screw results in better homogenization of the material and the development of frictional heat. The frictional heat added by the work of the screw is proportional to the square of the screw speed, if the screw speed is doubled, the heat added is increased by a factor of four.

Faster molding cycles are generally achieved with the screw-ram machines. polypropylene and the copolymers harden relatively fast when injection molded, and with the lower melt temperature possible with the screw-ram machines, the cycle can be shorten.

The physical properties of the items molded from the polypropylene and the copolymers on a screw-ram machines are better than those of identical items molded on a plunger type machine. Articles molded in the screw-ram machine contain fewer stresses because the mold cavity can be filled at a lower injection pressure. Reduced molding stresses results in parts better dimensional stability.

When polypropylene and copolymers are molded in colors, less time is required to change from one color to other when a screw-ram machine is used.

Polypropylene and copolymers behave in much the same way in processing operations, the conclusion drawn concerning one material generally apply to the other as well, except that the copolymers appear to be better suited than polypropylene for insulated runner molding on a screw-ram machine. The use of the preplasticating unit is not necessary, but it is advantage when polypropylene

Table 12 Tenite Polyethylene 4E31 Physical Properties

PROPERTY	UNIT	ASTM TEST METHOD	TYPICAL VALUE
Flow rate	g/10 min	D 1238L	4.5
Density	g/cubic cm	D 1505	0.90
Softening point	centigrade	D 1525	126
Deflection Temperature	centigrade Fahrenheit	D 648	50 122
Tensile strength at Yield	psi MPa	D 638 Type IV	3500 24.1
Stiffness in Flexure	100000 psi MPa	D 747	1.2 827
Flexural Modulus of Elasticity	100000 psi MPa	D 790	1.4 965
Rockwell Hardness	R scale ft-lb/in of notch J/m	D 785	67 5.0 267
Izod Impact Strength(at 73F) (at 0 degree F)	ft-lb/in J/m ft-lb/in	D 256	No break No break >25 >1335

and copolymers are molded in a plunger-type machine. By means of this unit, the high heat requirements of polypropylene and copolymers are partially provided before the material enters the cylinder. Therefore the cylinder can be maintained at a lower temperature than when it is supplying the entire heat inputs, and the possibility of hot spots is greatly reduced.

9.3.3 Molding Thin Sections

For the molding of small parts with wall thickness of 0.010 in. both polypropylene and the copolymers are best to use, because both have excellent moldability. In general, these plastics shows sharp decreases in viscosity at their melting pint. This allow them to flow in the mold cavities more rapidly than do most other thermoplastics.

9.3.4 Molding Thick Sections

The molding of thick sections from the general purpose polypropylene should be avoid because of the formation of the coarse crystalline structure in the article caused by slow cooling of the plastic. Articles with such a structure usually have a low impact strength. The toughness of articles molded of impact modified polypropylene or propylene copolymers is much less dependent upon rapid cooling, and these formulations may be better than general-purpose polypropylene for molding thick sections.

The wall thickness of the part is usually determined by the stiffness required in the molded piece and the material selected for the job. Because of the high stiffness of the polypropylene, a part to be molded of this plastic may be designed with thinner walls than ordinarily would be required with other polyolefins.

When it is necessary to mold a part with thick section, it is important to have the thick section near the gate with any reduction in thickness being made in the direction of the flow. This makes it possible to maintain effective pressure on the thick section of the part for a longer time without having excess pressure on the thin sections which are away from the gate. Gating into thick sections minimizes sink marks and results in less tendency to warping than does gating into thin sections.

9.4 Molding Techniques and Conditions Affecting Part

After the mold is constructed, the operating factors that affect the quality, quantity, and cost of the molded product must be determined. Although the quality, quantity and the cost of the product are the primarily dependent on the quality and the type of tooling and machine employed, proper molding techniques and the use of the optimum molding conditions have significant influence. The quality of the molded part is depend upon these factors: injection speed, injection pressure, clamping pressure, melt pressure, mold temperature, and cycle time.

9.4.1 Injection Speed

In case of polypropylene and copolymers, high injection speeds are used, because fast filling speed results in a relatively uniform temperature of the material as it fills the cavity. If the filling rate is slow, the material entering the cavity may cool much more rapidly than the subsequent material, resulting in an incomplete fill, lamination, and possible warpage of the part. This filling problem is associated with any thermoplastic , but it is not the problem in this case, which has a relatively high crystalline melting temperature and solidifies quickly in the cavity.

It may be necessary to reduce injection speed to control the uniformity of the flow and maintain a good surface finish when parts with thick cross sections are molded with small gates.

9.4.2 Injection Pressure

The injection pressure should be maintained at the minimum level required to fill the mold. Molding shrinkage may be reduced and sink marks minimized by increasing the injection pressure, but this results in packing the material into the mold cavity. Such packing may cause difficulty in ejecting the piece from the mold and warpage of thin sections. The ejection of part is more important than sink marks.

9.4.3 Clamping Pressure

Clamping pressure is the pressure needed to hold the mold closed against the opposing pressure exerted by the molten plastic under force of the injection and holding pressure.

The pressure transmitted to the mold cavity depends primarily on the type of injection unit used. For example, injection molding machines with any type of preplasticator in which the shooting ram works against molten polymer are very efficient in transmitting applied pressure to the cavity. Screw-ram machines are also very efficient, transmitting up to 90 percent of the applied ram pressure to the molten polymer in the mold cavity. A plunger type injection unit, in which the plunger acts against unmelted pellets, is less efficient in transmitting applied pressure to the mold cavity than are screw-ram machines or machines with preplasticators. Restriction in the nozzle, runner system, or gates retard the flow of molten polymer and limit the transmission of injection pressure to the cavity. A web gate can be used to good advantage when molding polypropylene and

copolymers because it gives the more effective area for transmitting pressure than other gates and will still freeze off when the flow stops.

9.4.4 Melt Temperature

The processing temperature for the polypropylene vary more with the characteristics of the processing equipment and its accessories than they do with the formulation, but in any given processing situation, the optimum temperature may vary with the flow rate of the material. The best melt temperature for the polypropylene is ranges between 380 to 450 degrees F.

As the melt temperature increase, there is a decrease in the stiffness and the impact strength of molded polypropylene and copolymers. The decrease in the stiffness caused by the increase in the melt temperature is greatest when the high injection pressure is used. Below certain melt temperatures, severe stresses in the molded part can occur with a resultant loss of impact strength.

At normal injection molding temperatures, around 450 to 470 degrees F., there is no significant difference in the deflection temperature caused by the changes in melt temperature. At extremely high temperatures, about 500 to 550 degrees F., an increase is noted. High melt temperature with long residence time at melt temperature may result in increased flow rate and reduced toughness. It is desirable that the shot size utilize one-half or more of the cylinder capacity to limit melt residence time.

9.4.5 Mold Temperature

Close control of mold temperature is important in molding any thermoplastic, but it has increased significance in the molding of the polypropylene and copolymers because of the highly crystalline nature of these plastics. Mold temperature affects the properties of copolymers less than it does those of polypropylene, and

a relatively tough part can be molded from Tenite polyallomer copolymer with only limited mold cooling. Mold temperature upto 90 degrees F. are usually used.

Tenite polyallomer copolymer crystallizes more slowly than the polypropylene, and a portion of the crystallinity in the molded part forms after the part is removed from the mold. It is for this reason that the mold temperature has less effect on the properties of polyallomer than it does on the properties of general-purpose polypropylene. In molding polypropylene and copolymer parts, it is usually desirable to obtain maximum impact strength rather than maximum stiffness. This indicates that low mold temperatures should be used, normally in the range of 30 to 60 degrees F. A cold mold cools the material rapidly and causes the formulation of a fine crystalline structure.

In molding articles of heavy cross sections, where high mold temperatures may be necessary, it may be advantageous to cool the articles in an ice water bath immediately after ejecting them from the mold. This allows the article to be ejected while still hot and thus shorten the cooling portion of the molding cycle. Cooling the parts in ice water also achieves the quick quenching necessary for good impact strength, and it hardens the surface sufficiently to prevent sink marks from forming.

9.4.6 Cycle Time

Cycle time is largely dependent on the thickness, machine conditions, machine heating capacity, and injection capacity. The overall cycle time can vary from 5 seconds for thin articles to 60 seconds or more for thick articles.

CHAPTER 10

CONTROL OF PROCESS

10.1 Introduction

All molding machines with little effort are capable of providing useful melts that go into molds and provide salable products. On the average, at least half of the costs in plastics processing are incurred in raw materials and services; wages, utilities and capital costs account for the rest.

Thus it is important to purchase the raw materials at favorable prices, to have them delivered punctually, to use as little as possible, and to ensure that their quality remains constant. Savings may be effected by judicious selection of the form in which material are supplied.

The system for ordering materials depends on the production program. It may be based on requirements, stocks, or agreed-upon deadlines. Costs can be saved by finding out the qualities that can be supplied on the most favorable terms. Decrease in the price effected by purchasers larger amounts must be balanced against the extra cost for the storage and the larger amount of tied-up capital; a certain amount must represent an optimum. Purchasers must also allow for delivery times. Frequently materials in the natural color can be supplied direct from stocks. The next step after this process is checking of the material.

10.2 Checking of the Material

An important factor in the production of the parts is the quality of the raw materials must always conform to specification. Certain properties must be checked when the goods are received. In view of the wide variety of the applications for plastic articles, a testing schedule of general validity cannot be submitted here. Each case must be treated individually.

Over the years, many hours have been devoted to designing methods for testing materials to develop values for their properties. These tests, conducted under procedures established by organizations such as the American society for testing and materials (ASTM), are the means of extracting basic knowledge about materials. Although raw materials of constant properties are essential for the high-quality moldings, they do not suffice for this purpose by themselves. In particular, mistakes in the processing could adversely effect the properties. If possible allowance must be made for this potential problem in the testing schedule.

The first task in checking goods received is to make sure that they conform to type. In other words, they must be checked to ensure that they agree with samples of former deliveries. This check includes examination for contamination, and is followed by the specific tests such as simply determining bulk density. Often samples are sent in advance of materials dispatched in the tankcars or large containers. In this case, statistical rules must be observed in taking the random samples.

The preliminary check must proceed without loss of time; so rapid test with specific aims are frequently used. Since injection molding has been caught up in the automation trend, it is feasible for checking the goods received to become part and parcel of the actual production process. However, this entails that any deviation from standard must remain within narrow limits. For technical and economic reasons, this adaptive process control, as it is called, is still a long way from being realized.

10.3 Compounding and Coloring In-Plant

Compounding or mixing is an important stage in the production of the raw materials. The way it is performed can effect injection molding, especially if the

compound is in the form of a powder, and the ingredients are not mixed together until shortly before molding.

Great significance has been attached to adding all kinds of masterbatches, for example. There are color masterbatches, reinforcing fiber masterbatches, flame-retardant and anti static masterbatches, and masterbatches containing foaming agents and other additives. Since the importance has been recognized of what are known as plastics alloys, which widen the field of application of thermoplastics, different pellets or powders are also mixed with one another.

A distinction based on the stirrer speed is drawn between gravity mixers and stirrers. The peripheral velocity in slow stirrers is 30 ft/sec (10 to 50 m/s).

In-plant blending of the molding compounds offers some advantages. It dispenses with some of the fabrication costs and potential problems due to heat history, and greatly reduces inventories. Purchasing one type in bulk reduces the cost of the raw materials. Production can be made very flexible to cope with the small amounts and special wishes. One of the most important tasks in the injection molding factory is in-plant coloring. The advantages are obvious: saving costs incurred by the higher prices of colored grades, a wider selection of colors, adaptability, and reduced inventories. However, these advantages are balanced by the responsibility of selecting suitable colorants. Moreover, the colorants must not impair the properties of the molding compound. At any rate, the demand imposed on the quality of the shades and their reproducibility from one machine to another and from one batch to another can never be so severe as those imposed on molding compounds supplied by the raw material manufacturer. The cycle may become longer, and the shrinkage may change. Consequently, the workers entrusted with in-plant coloring are chiefly responsible for its quality. Formerly, mixers were set aside in special rooms for in-plant coloring. They are now being supplemented by devices that allow coloring on the injection molding

machine. They can proportion as many as three types of colorants, the molding compound in the natural color, and the regrind, and are usually fitted with the mixer. The colorants are in the form of the pellets, ground masterbatches, free-flowing and non free-flowing pigment powders, pigment dispersions, and pumpable liquids. Great values must be attached to their dispersibility.

The quality of the coloration obtained with in-plant techniques depends not only on the proportioning and mixing in the feeding device but also on the plastification in the injection molding machine. Frequently, screw with the mixing attachments in the metering section or with the static mixers connected in the series behind the metering zone are indispensable.

In-plant blending of the virgin plastic with granulated or recycle plastics is important to proper control. If not controlled, performance of the part can be below requirements.

10.4 Production Control of Quality

Quality control is a complex task in injection molding. The quality and the serviceability of a molding depend on many factors, starting from the raw materials and embracing the processing and application conditions.

Quality control begins with the design of the part, design of the mold, and capability of the injection molding machine. The number of the cavities, the type of the location of the sprue, the size of the machine, the allowance to be made for inserts, demolding, finish, and the tolerances laid down, are all factors that decide the quality and govern the price. In the early mold design stage, the test to be adopted for quality control should already be decided upon and drawn up in the form of the checklist that will be accepted by the customer concerned. The optimum injection conditions are determined in the trial runs and noted in a

report. The moldings thus produced are tested according to the checklist. The acceptance tests for the raw materials are a part of quality control.

The live production run is usually controlled by continuous visual inspections of the moldings and by checking their weight and a few dimensions. Measuring the dimensions at this stage is only of a relative value because processing shrinkage is not always completed after the moldings have cooled. This applies particularly to partially crystalline molding compounds.

10.5 Economic Control of Equipment

In view of continuously rising costs, the main consideration in investing capital must be the ratio of earnings to costs. Production aids can make a considerable contribution to reducing costs. The most important are those required for feeding the raw material, deflashing, regrinding and recycling scrap, sorting the molding from the sprues, demolding, stacking, packing, automatic machining, and bonding with adhesives. The only item that does not rise in cost is the machine performance. There are always new machines that will provide lower cost to melt the plastics.

Factors to be considered in the acquisition of new injection molding machines are the criteria set up by the intended production program. For the injection molding of the packaging, the main factors are the injection rate, the dry cycle time, the plastification rate, and the price of the machine. As opposed to this, the quality of the melt, process control aspects, and the clamping force are the factors that predominate in the production of machine precision parts.

Other requirements that are imposed on an injection molding machine for economical running are favorable starting-up characteristics, constant production characteristics, ease of operation, ease of the retooling, and a long life. Savings can be achieved in tooling by standardizing the platens, the radii of curvature, the

fitting and the electrical circuit. Machinery cost can be reduced by parts that do not require maintenance. This applies particularly to the hydraulic system.

Practically any step involved in processing the plastic contributes to cost and can easily be evaluated with respect to cost reduction. Consider, for example, when you should replace your machine as well as upstream and downstream equipment. Various methods can be used to replace old equipment. In United States today, a lot of molders are losing money with old equipment, and they do not even know it. Not only are the new machines more productive; they also create less waste, use less energy, and are smaller, quieter and safer.

Savings may also be possible in costs for fresh water and effluents, which have increased rapidly. There is generally a shortage of water in a period of dry weather, and water consumption in the factories is growing as a result of increasing mechanization. Consequently, many injection molding factories have their own cooling water supplies. The main types are:

1. Open circuit water cooling systems with an evaporating type cooling tower.
2. Closed circuit water cooling system with compression type refrigeration machines.
3. Composite systems.

Open circuit cooling systems operating exclusively with cooling towers were very popular in the past but now lost their efficiency. As a result of evaporation and the slime formation, 1.5 to 2.5 percent of the water circulated is lost and must be replenished. The temperature and the humidity of the ambient air impose limit on the temperature that can be attained by the cooling water. At most, the temperature of the cooling water can be reduced to a value of 3 degree centigrade above the wet bulb temperature. The compressor type refrigerating machines in the closed circuit systems operate with the air cooled condensers.

Reciprocating machines and the turbocompressors predominate. The main refrigerant is liquified fluorohydrocarbon under pressure.

Combination of the open system and the closed systems also operates with evaporation type cooling towers. Normally, the temperature of the cooled water in the closed refrigerating machine circuit is between 5 and 20 degree centigrade. This water is used for cooling The mold. A second system of pipes carries the water that is cooled by flowing over the cooling tower. This water is used for the condenser of the refrigerating machine and for the hydraulic system. The twin-circuit system saves great amounts of energy because it can function as a single-circuit system in winter with the evaporation-type cooling tower. In summer, it is refitted as a twin-circuit system.

10.6 Machine Save Energy

It is important to evaluate how much energy a machine requires for its operation. There are two types of machines, those that require a great deal of energy and those that require less.

Injection molding is one of the most energy intensive methods employed for converting plastics resin to a finished product. It requires not only the energy used by the machine to drive the motor or motors for the hydraulic power, but the energy to the heater bands to melt the resin, as well.

Then there is the problem of removing the heat that is generated in the hydraulic system b y using water in the heat exchanger, and water is also needed to cool the mold to remove the heat from the plastic. This water can be from the city system, and depending on the machine size and mold and the water temperature available, as much as 20 to 30 gallons per minute could be required, thus creating a sizable water bill. Most plants have acquired their own wells, or closed systems using cooling towers, chillers, and the like. These require pumps

and motors, plus, in the case of chillers, compressors as well. Machine grinders are quite often used, plus material handling equipment. In all a lot of energy is used for the process.

It is estimated that the cost of the energy is doubled in the next five years. This being the case, the molder is faced with two problems. First is the cost of the new machine, he should buy the most energy efficient machine available. This long term investment, so that the price alone or any other single reason is not justified when the long term use of energy is considered. Also, a machine that is not energy-efficient may be difficult, if not impossible, to resell a few years later. No one can go out and replace all of his machines with energy-efficient ones, a situation that leads to the second problem: we must, if possible, reduce the energy used on present equipment.

Energy consumption in the molding machine is directly related to the hydraulic pressure used. The higher the pressure, the more power, and thus the more energy needed. So the basic approach is to determine how to reduce the pressures required to do the job.

First of all, consider the clamp. The more tonnage that is required to lock up the mold, the higher the hydraulic pressure must be to accomplish this. Whether we are talking about a hydraulic ram machine or a toggle machine, the problem is the same. Basically we are trying to hold the mold closed against the force of injection to prevent flashing. The first consideration is the mold. Is the mold base relieved to minimize the area of the mold that must be clamped to ensure the good shut-off. This is a relatively inexpensive adjustment which would allow using less clamp. Less clamp tonnage translates into less energy used, but also improves running conditions, as the vents are more effective. So spend a few dollars on the mold to ensure good operating conditions. The greatest use of the energy occurs at the injection end. There energy is used to produce the melt and

the force it into the mold. The heaters bands draw electrical energy to melt the plastic along with the screw drive, which provides some heat to the plastics through shear. putting the plastics into the mold requires high pressure and a large pump capacity.

There are quite a few ways to help reduce energy cost in this area. First, consider the screw recovery or plasticating. Probably the most efficient way to run the screw is with about 60 to 70 percent of the heat being provide by the heater bands and the remaining 30 to 40 percent by shear. To accomplish this one needs to know something about the screw and how it works in order to arrive at a heat profile suitable to the resin being processed and the rate at which it is processed.

A starting point would be to set the rear zone of heat at about 50 degrees above the softening point of the resin to be run, the center zone about 50 degrees Fahrenheit above the front zone, and the front zone at the stock temperature that one desires to run at. Watch the screw drive pressure during the recovery. It should be at about 50 to 65 percent of the maximum available. It is below 50 percent, no shear heat is being used and the mix of the melt is not very good, particularly if coloring is being used. Above 65 percent, too much of the heat is being put in through shear, a condition that is not energy-efficient.

Heater bands have received a lot of publicity recently as a possible energy saving source. The bands touted as energy savers are the ceramic element bands with one-half inch of insulation over them. The only change that we made was in the heat profile, to maintain the target temperature when we change heater band conditions.

We found that a heat sink problem occurs, in that the insulation directly on the heater band does not allow for the modulation of the heat at the surface, which is greater than at the thermocouple, so this greater heat has no place to go

but down through the steel to the plastic. The full cover with uninsulated bands provides an oven effect which eliminates that condition.

The variation in the melt temperature, on amorphous material particularly, can affect molding conditions to the point of providing slight non-fills or sink to the slight flashing due to viscosity change with the temperature change.

Our recommendation is that uninsulated bands with a full cover be used as the most energy-efficient arrangement, which provides the best control over the melt. Testing of the blanket over the cover is not complete, but this idea is good for air-conditioned operation.

The force required to put the plastic material into the mold consumes most energy. Contributors to this problem are the viscosity of the melt, the size of the gate, the setting of the pressures, and the speed of fill, as well as the duration of the boost or delay unload. The viscosity of the melt must be carefully control to get the best quality of the melt possible. The size of the gate is another matter. Gate sizes are usually smaller than is necessary because it is easier to make it big as required. Once the mold is filled it is impossible to change the size of the gate.

A very small change , in thousands of an inch, can have significant bearing on the cross-sectional area of the gate. for example, going from a 0.040-in. gate to a 0.05-in. gate, results in a 56 percent increase in area. That would have a decided effect on the pressure required to fill the mold. It could also mean a reduction in melt temperature, which translates into faster cycle due to there being less heat to remove from the mold.

So gating is the significant part of the play in the energy used.

1. Do not try to run a tool that is not in good condition. A few hundred dollars spent on tool maintenance can save thousands of dollars spent on wasted energy.

2. Do not use more clamp than necessary.
3. Learn how to use the screw to best advantage. Talk to your supplier, and get his recommendations.
4. Use as low an injection pressure as possible to reduce the pressure required.
5. Reduce melt temperature if the gate size will let you.

You save energy on melt preparation and on removing it from the mold, and improve cycle time as well. By means of reduction in melt temperature we can control lot of problems.

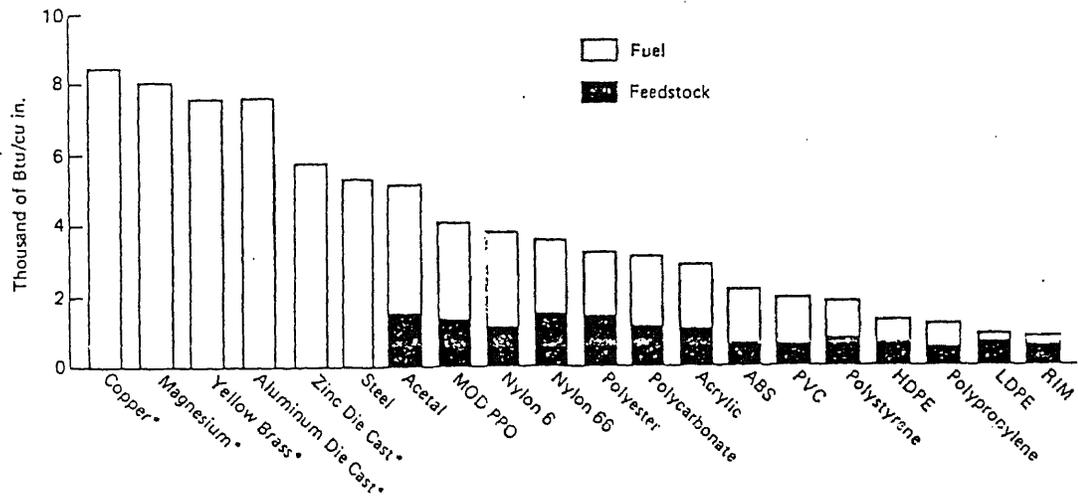
10.7 Plastics Save Energy

There are always improvements to be made in the machines and equipment in the plant whereby energy saving can be made with the net savings in the total production costs. But sometimes equipments can be made more energy-efficient and a condition during molding will cause a total increase in cost.

But if we study the relationship between the energy savings vs. the use of the practically any other materials, plastics conserve energy in the significant ways. Energy is saved in the service life of the plastics product. Energy is also saved in shipping and maintenance, since plastics are lightweight and require less fuel for shipping and are inherently inert to chemicals, rot, mildew, corrosion, and hostile environments. Another important aspect of their use is that as new markets for plastics are developed, new ways to save the energy are found in all phases of the manufacturing process and in performance.

Of the many uses of the petrochemicals, the production of the plastics materials is the most ingenious. The versatility of these long-chain macromolecules of basic elements combined to make diversified products in testimony to the imagination and talents of the industry. Since the injection

Table 13 Energy Requirement for Different Materials



molding process is a high high pressure high speed process, it is reasonable to assume that the machines should be saved.

10.8 Machine Safety

Thus machine safety is a must to ensure operator safety. There are procedures that outlines how to operate your machines and other plant operations, including plastic storage procedures, moving plastics around the plant. Operators of machines should consider steps to be taken that will ensure personal safety. An example is in the proper lockout of the machine's electrical circuit. Properly locking out the machine's electrical circuit before starting repairs protects the maintenance workers from accidental start-ups, which could cause severe injury. The National Safety Council offers the following steps for the proper lockout procedures:

1. Shut down all possible switches at the point of operation, then open the main disconnect switch.
2. Snap your own lock on the locking device. An ordinary padlock can be used for most electrical lockouts.
3. Check the lockout device to ensure the switch can't be operated.
4. Place a name tag on the shank of the lock to indicate that the machine has been locked out.
5. Notify the supervisor when the repair work has been completed. Only he should give the go ahead to remove your lock.
6. Take off the name tag and remove the lock.

CHAPTER 11

DIE TOLERANCES FOR PLASTIC MOLDING

11.1 Mold Shrinkage

The dimensions to which a cavity and core should be fabricated in order to produce a part of desired shape and size is called shrinkage or tolerance. The usual way to decide the amount of shrinkage is to consult the data supplied by the manufacturer. The supplier's information is obtained from a test bar molded according to an ASTM standard. The test bar is molded at specific pressure, mold temperature, melt temperature, and cure time. The thickness of the test bar is normally 1/8 in. However, molded parts are very rarely produced under conditions and sizes that are the same as or even similar to those used for test bars.

For precision parts with close tolerance dimensions, shrinkage information from the supplier is not enough but it is very feasible as a guide. We must familiar with the factors that influence shrinkage so that we may arrive at more exact dimensions for a specific part. According to compiled data, shrinkage is a function of mold temperature, part thickness, injection pressure, and melt temperature.

Shrinkage is influenced by the cavity pressure to a very large degree. Depending on the pressure in the cavity alone, the shrinkage is vary as much as 100 percent. Part thickness will cause a change in shrinkage. A thicker piece will have the shrinkage value on the high side of the data, whereas a thin one will have a lower shrinkage value. The mold and the melt temperature also influenced shrinkage. A cooler mold results in less shrinkage, whereas a hotter melt will cause more shrinkage, compared to the supplier's information. The longer the part in the cavity, the closer the part comes to mold dimensions, which

means a lessening of shrinkage. Openings in the part will cause variation in shrinkage from section to section because the cores making these openings act as temporary cooling blocks, which prevent change in the dimension while the part is solidifying. A relatively large gate permit higher cavity pressure buildup, which brings about a lower shrinkage.

11.2 Categories for Shrinkage Problems

1. Amorphous material with shrinkage rate of 0.008in/in. or less have readily predictable shrinkage, which is not difficult to adjust with a molding parameters such as cavity pressure and mold or melt temperature, or with the cycle.
2. Parts made of crystalline materials with high shrinkage, but which are symmetrical and suitable for center gating, will also have a readily predictable shrinkage, adjustable with molding parameters.
3. Parts made of materials with a high shrinkage rate that are symmetrical but cannot be center-gated, may approximate a center-gate condition if multiple gating close to the center is possible. In this case the prediction of shrinkage is somewhat more difficult but still presents a chance of success.
4. The major problem exists with materials that have a high shrinkage rate of about 0.015 to 0.035in./in.

In most of these case, the material suppliers either show nomographs in which all contributing factors are drawn and coordinated to supply reasonably close shrinkage information, or they point to examples with actual shrinkage information and molding parameters so they can be used comparative interpolation. With most high-shrinkage crystalline materials, when the material is side-gated, a large shrinkage is occur in the direction of the flow an the smaller one perpendicular to it. One more way of establishing accurate shrinkage data is

prototyping. In this method a single cavity is built, and the critical dimensions are so calculated that they will allow for correction after testing, by providing for metal removal. The test sample should be run for at least half an hour and under the same conditions as a production run. Only the last half-dozen pieces from the run should be used for dimensioning.

It is best to make the measurements after a 24-hour period at room temperature. However with crystalline thermoplastics such as acetal, nylon, thermoplastic polyester, polyethylene, and polypropylene, the ultimate shrinkage may continue for days, weeks, months, or even a year. The shrinkage noted 1 hour after molding may be only 75 to 95 percent of the total.

The reason for post molding shrinkage is that there is a molecular rearrangement and stress relaxation going on until equilibrium is attained, at which point shrinkage stops, both the molecular arrangement and the stresses are brought about by molding conditions. The conditions that are most favorable for reaching the ultimate shrinkage in shortest time are relatively high mold temperature and a lower rate of freezing of the material. Each material has its own rate of postmolding shrinkage as a function of time.

CHAPTER 12

TROUBLE SHOOTING PROGRAM

12.1 Introduction

Molding cycle problem analysis is an important field in plastic injection molding process. It is very important to catch the exact approach in eliminating all sort of problems. Therefore on the basis of different suggestion from many companies a software is prepared which deals with the trouble shooting in molding cycle problems. In this software possible remedies are given against the problems.

12.2 Trouble Shooting

Here practical possible remedies have been classified according to:

1. Material
2. Mold
3. Molding cycle
4. Machine performance

In practice most of the faulty parts usually result from problems which are connected with these three categories:

1. Premolding
2. Molding
3. Postmolding

These include problems like contamination, color, static dust collection, painting, and vacuum metalizing. We discuss only those problems which are associated with molding cycle. i.e.

1. Fill time
2. Packing time / Rate
3. Cooling time

4. Ejection time
5. Open time
6. Mold time
7. Sprue and runner design
8. Gate size and location
9. Section thickness
10. Length of flow path

The three major elements in the molding operation are;

1. Injection molding machine
2. Mold
3. Material

The performance of these three major elements is influenced by three major variables, i.e.

1. Time
2. Pressure
3. Temperature

12.3 Approach Should be Exact

Good quality control is necessary for finding any fault. Quality control is associated with each and every step of operation, from raw material ordering to shipping of the finished good.

The cause of the problem may be obvious, and the problem corrected by the adjustment in three major variables. If the area of the difficulty is not apparent then each set of adjustment variables must be examined and corrections made where necessary. When a molder is starting up a new mold using a material on which he has certain data, he uses past experienced on similar molds and materials to set up an approximate cycle. If the moldings are not perfect on this

cycle, he will try to vary the pressure, temperature, and the time sequences by adjusting the machine conditions until he obtains good result i.e. according to his expectations.

In case if acceptable pieces are not produced after adjustment of machine conditions, then the design of the mold should be examined. It is obvious that any change in the mold design can affect the temperature, pressure, and time sequences.

Most molding problems are solved by varying the machine conditions and by changing the design of the mold. But some problems remain unchanged, therefore in such a case fault and possible solutions may be found by examining polymer variables such as:

1. Flow characteristics
2. Thermal properties
3. Granulation

12.4 Problem Solving

For problem solving there are some basic rules:

1. Planning
 2. Molding conditions
 3. Change one condition at a time
 4. Allow sufficient time at each change
 5. Keep an accurate log of each change
 6. Check housekeeping
 7. Get approach to problem area, i.e. the type of problem whether it is belong to the machine, mold, operating conditions, material, part design or management.
- for example:

- * Change the material: If the problem remains the same, it probably is not the material.
 - * Random trouble is probably a function of the machine, the temperature control system, or the heating bands.
 - * If the problem appears in about the same position of a single-cavity mold, it is probably a function of the flow pattern and the system from the front of the plunger through the nozzle, sprue, runner, and gate.
 - * If the problem occurs in the same cavity or cavities of a multi-cavity mold, it is in the cavity or gate and runner system.
 - * If machine operation malfunctions, check hydraulic and electric circuits.
- (8) Set up a procedure to "Break in" a new mold.
- * Obtain samples and molding cycle information, if the mold is new to the shop but has been run before.
 - * Clean the mold
 - * Visually inspect the mold
 - * Check out actions of the mold
 - * Install safety devices
 - * Open the mold and inspect it
 - * Dry cycle the mold without injecting material
 - * Record operating information

12.5 Conclusion

The key to understanding trouble shooting is to gain as complete as possible a knowledge about the machine i.e. what it is doing to the plastic, then about plastic i.e. what the plastic is doing to the mold.

REFERENCES

1. Arnold, Lionel. 1975. "Introduction to Plastics." The Iowa State University Press, AMES.
2. Baird, Ronald., and Donald Jambro. 1971. "Industrial Plastics." Goodheart Willcox, Company. Inc., New York.
3. Becker, Walter. 1979. " Reaction Injection Molding." Van Nostrand Reinhold, Company. New York.
4. Bown, John. 1979. "Injection Molding of Plastics Components." Mc Graw-Hill Book Company.
5. Briston J H., and Lionel Arnold. 1968. "Introduction to Plastics." Richard Clay Ltd. Great Britain.
6. Bruins, Paul. 1978. " Basic Principles of Thermoforming." Gordon and Breach, Publishers, Inc., New York.
7. Crosby, Edward. 1965. " Practical Guide to Plastics Application." Cahress Book Division of Cahress Publishing Company, Inc., Massachusetts.
8. Driver, walter. 1979. " Plastics Chemistry and Technology." Van Nostrand Reinhold, Company. New York.
9. Dubois, Harry. 1967 " Plastics." Reinhold Publishing corporation. New York.
10. Dym, Joseph. 1987. " Injection Moild and Molding." Van Nostrand Reinhold, Company. New York.
11. Furness, ralph W. 1968. " The Practice of Plating on Plastics." Clare O' Molesey, Ltd. Surrey.
12. Jambro, Donald. 1976. " Plastics." Prentice-Hall, Inc., NJ.
13. Karz, Sylvia. 1978. " Plastics Design and Materials." Macmillan Publishing Company, Inc., New York.
14. Newman, Thelman. 1972. " Plastics as Design Form." Chilton Book Company. New York.
15. Pye, R G W. 1983. " Injection Mold Design." Godwin Publishers, Inc., New York.
16. Seymour, William. 1968. " Modern Plastics Technology." Reston Publishing, Company, Inc., Virginia.
17. Simonds, Herbert. 1965. " A Concise Guide to Plastics." Reinhold Publishing, Corporation. New York.
18. Smith, W Mayo. 1964. " Manufacture of Plastics." Reinhold Publishing, Corporation. New York.

19. Swanson, Robert. 1965. " Plastics Technology." McKnight and McKnight, Publishers, Inc., ILL.
20. Sweeny, Melvin. 1979. " Introduction to Reaction Injection Molding." Technomic, Publishing Company, Inc., CT.
21. Weir, Clifford. 1975. " Introduction to Injection Molding." Society of Plastics Engineers. Inc., Conn.
22. Wendle, Bruce C. 1978. " Engineering Guide to Plastics Plant Layout and Machine Selection." Technomic, Publishing Company, Inc., CT.