

CONCEPTUAL DESIGN OF TWO-PLATE INJECTION MOLD FOR UMP'S
KEYCHAIN

AZHAR BIN YUSOP

A report submitted in partial fulfillment of the requirement for the award of degree of
Mechanical Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

MAY 2008

PERPUSTAKAAN UNIVERSITI MALAYSIA PAHANG	
No. Perolehan 037955	No. Panggilan TP 1150 -A94 2008 rs K2
Tarikh 02 JUL 2009	

ABSTRACT

UMP's keychain is a small product made from plastic same as UMP (University Malaysia Pahang) logo. Demand on plastic product in this country is very tremendous because plastic product has better in quality, design and appearance than any material product. To produce better quality of plastic product, it needs to have some processes and most important is initially in design stages. High quality plastic product can be produced using injection molding process that required good design of mold. In design the plastic mold the most important thing is get the suitable size runner and gate system. This project is using Moldflow Mold Adviser software in order to get achieve optimum size runner and gate system for two-plate injection mold for UMP's keychain

ABSTRAK

Rantai kunci UMP ialah satu produk kecil yang di perbuat dari plastik berbentuk logo UMP (Universiti Malaysia Pahang). Pada dasarnya permintaan produk plastik di negara ini sangat menggalakkan kerana produk plastik adalah setanding dengan produk yang dihasilkan dari bahan yang lain malah produk plastik juga lebih cantik dari segi rupa bentuk serta bermutu. Maka dengan itu untuk menghasilkan produk plastik yang bermutu, produk plastik yang ingin dihasilkan perlu melalui beberapa proses yang sepatutnya terutamanya yang penting sekali ialah pada proses permulaan yang melibatkan proses reka bentuk. Produk plastik yang baik dapat dihasilkan dengan menggunakan proses *injection molding* yang memerlukan reka bentuk acuan yang baik. Untuk mereka bentuk acuan plastic, yang paling penting adalah mendapatkan ukuran *runner* dan *gate* sistem yang sesuai. Kajian ini menggunakan perisian Moldflow Mold Adviser untuk mendapatkan ukuran *runner* dan *gate* sistem yang optima untuk 2-plate suntikan acuan untuk rantai kunci UMP.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	TITLE PAGE	i
	SUPERVISOR DECLARATION	ii
	STUDENT DECLARATION	iii
	DEDICATION	iv
	ACKNOWLEDGEMENTS	v
	ABSTRACT	vi
	ABSTRAK	vii
	TABLE OF CONTENTS	viii
	LIST OF TABLES	xii
	LIST OF FIGURES	xiii
	LIST OF SYMBOLS	xvi
	LIST OF APPENDICES	xvii
1	INTRODUCTION	1
	1.1 Project Background	1
	1.2 Problem Statement	2
	1.3 Project Objective	2
	1.4 Project Scope.	2
2	LITERATURE REVIEW	3
	2.1 Introduction	3
	2.2 Injection Molding	3
	2.2.1 Raw Material	4

2.2.2	Injection Molding Cycle and Process	4
2.3	Part Design Guideline for Injection Molded Plastic Part	5
2.3.1	Wall Thickness	5
2.3.2	Corner, Fillet and Radii	6
2.3.3	Rib	7
2.3.4	Bosses and Gussets	8
2.3.5	Taper and Draft Angles	9
2.4	Injection Mold Design.	9
2.4.1	Type of Injection Mold	10
2.4.2	Standard Mold Assembly	10
2.4.2.1	Mold Part Name and Functions	11
2.4.3	Cavity and Core	13
2.4.3.1	Construction of Cavity section.	14
2.4.4	Passageway of Molten Resin	15
2.4.4.1	Sprue	15
2.4.4.2	Runner	16
2.4.4.3	Gate Design	17
2.4.5	Mold Temperature Control	19
2.4.6	Venting	19
2.5	Mold Design Procedure	20
2.5.1	Decision of Number of Cavity	20
2.5.2	Decision of Arrangement of Cavities	20
2.5.3	Decision of parting line (P.L)	21
2.5.4	Decision of Under Cut Treating Method	21
2.5.5	Decision of ejecting method.	22
2.5.6	Decision of Cavity and Core Construction	22
2.5.7	Decision of Runner System	23
2.5.8	Decision type of gate	23
2.5.9	Decision of Temperature Control Method	23
2.5.10	Decision of Cavity and Core Fixing Method	23
2.5.11	Decision of Mold Material and Surface Treating Method	24

2.5.12	Decision of Mold Dimension	24
2.5.13	Plotting of layout drawing	24
2.6	Mold flow software	25
2.6.1	Moldflow mold advisor(MPA)	26
2.6.1.1	Analysis Capabilities	26
2.6.1.2	Advantages of Moldflow Analysis	27
3	METHODOLOGY	29
3.1	Introduction	29
3.2	Drawing of Product	31
3.3	Material Selection	32
3.4	Analysis Sequences	32
3.4.1	Best Gate Analysis	33
3.4.2	Molding Window	33
3.4.3	Sink Mark analysis	33
3.4.4	Runner balance analysis.	33
3.4.5	Fill and Cooling analysis	34
3.5	Analysis by using Moldflow Software(MPA 8.1)	34
3.6	Design the Injection Mold of UMP keychain	34
4	RESULTS AND DISCUSSION	37
4.1	Introduction	37
4.2	Analysis of Best Gate Location	37
4.3	Result Analysis on First Gate Location	38
4.3.1	Molding Window Analysis	39
4.3.2	Sink Mark Analysis	40
4.3.3	Runner Balance Analysis	41
4.3.4	Plastic Filling Analysis	43
4.3.4.1	Confidence of Fill	43
4.3.4.2	Cooling Analysis	48
4.4	Result Analysis on Second Gate Location	50

4.4.1	Molding Window Analysis	50
4.4.2	Sink Mark Analysis	51
4.4.3	Runner Balance Analysis	52
4.4.4	Plastic Filling Analysis	54
4.4.4.1	Confidence of Fill	54
4.4.4.2	Cooling Analysis	59
4.5	Discussion	60
4.5.1	Analysis of Gate Location	60
4.5.2	Molding Window Analysis	61
4.5.3	Sink Mark Analysis.	61
4.5.4	Runner Balanced Analysis	61
4.5.5	Filling Analysis	62
4.5.5.1	Fill Time	62
4.5.5.2	Confidence of Fill	63
4.5.5.3	Flow Front Temperature	63
4.5.5.4	Skin Orientation	65
4.5.5.5	Weld Line Prediction	65
4.5.5.6	Air Trap Estimated	66
4.5.5.7	Cooling Analysis	66
4.5.6	Conclusion between First and Second Gate Location	66
4.6	Design of Two-Plate Injection Mold for UMP Keychain	67
5	CONCLUSION AND RECOMMENDATION	71
5.1	Conclusion	71
5.2	Recommendation	72
	REFERENCES	73
	APPENDICES	74-82

LIST OF TABLES

TABLE NO.	TITLE	PAGE
2.1	Type of runner layout	16
2.2	Types of gate	17
3.1	Typical Injection Molding Conditions for ABS material	32
4.1	The changes advised for first gate location	42
4.2	Risk of the part filling base on color	44
4.3	Summary result filling analysis at the first gate location	48
4.4	The changes advised for second gate location	53
4.5	Risk of the part filling base on color	55
4.6	Summary result filling analysis at the second gate location	58
4.7	Runner balance advisor for two difference gate location	62
4.8	Comparison of result analysis result for first and second gate location	67

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
2.1	Recommended change at the intersection of thick/thin region	6
2.2	Proper designs of corners in a plastic part	7
2.3	Guideline for design of rib respect to wall thickness	7
2.4	Free standing boss with four gussets	8
2.5	Free standing boss with two gussets	8
2.6	Draft on the inside and outside surface of part should be equal and parallel	9
2.7	Standard mold	11
2.8	Cavity and core	13
2.9	Mold without core	13
2.10	Direct profile types	14
2.11	Insert type	15
2.12	Guideline for design of sprue	15
2.13	Cross-section design of runner	16
2.14	Recommended depth and pitch of mold cooling channel	19
3.1	Flow Chart	30
3.2	UMP keychain.(front view)	31
3.3	Ump keychain(side view)	31

3.4	Ump keychain (Top view)	31
3.5	Direct profile types.	35
3.6	H-Type runner layout.	35
3.7	Parting line	36
4.1	Analysis of the best gate locations for the Ump's keychain	38
4.2	Analysis of the best gate locations for the Ump's keychain	38
4.3	First gate location	39
4.4	Equivalent injection time and melting temperature	40
4.5	Sink Mark estimated for the UMP keychain by using edge gate	41
4.6	Sprue (A B), Runner (C D) and Gate (E) using for analysis at first gate location	42
4.7	Fill time by using the original design of sprue runner and gate	42
4.8	Fill time by change the dimension of runner and gate as per moldflow advised	43
4.9	Confidence of filling at the first gate location	44
4.10	Confident of fill after change mold and melt temperature	45
4.11	Temperature of flow front	45
4.12	Orientation of skin result from first gate location	46
4.13	Weld line prediction	47
4.14	Air trap estimation by using edge gate in two-plate mold	47
4.15	Design of cooling system.	48
4.16	Surface temperature variance by using edge gate at first gate Location	49
4.17	Cooling time variance by using edge gate at first gate location	49
4.18	The cooling quality by using edge gate at first location.	50

4.19	Second gate location	50
4.20	Equivalent injection time and melting temperature.	51
4.21	Sink Mark estimated for the second gate location analysis	52
4.22	Gating system for second gate location analysis	53
4.23	Fill time from original gating system for second gate location analysis	53
4.24	Fill time by change the dimension of runner and gate as per moldflow advised	54
4.25	Confidence of filling at the second gate location	55
4.26	Flow front temperature by using edge gate at second gate location	56
4.27	Skin orientation on the UMP keychain	56
4.28	Weld line prediction by using edge gate at second gate location	57
4.29	Air trap estimation by using edge gate at second gate location	58
4.30	Temperature Variance by using edge gate at second gate location	59
4.31	Cooling Time Variance by using edge gate at second gate location	59
4.32	Cooling Quality by using edge gate for second gate location	60
4.33	Flow front temperature result displays	64
4.34	Complete assembly of two-plate mold	68
4.35	Fixed half mold (Cavity side) side view of two-plate mold	68
4.36	Moving half mold (Core side) side view of two plate mold	69
4.37	Plan (Top) view of complete assembly two-plate mold	69
4.38	Side view of complete assembly two-plate mold	70

LIST OF SYMBOLS

ABS	-	Acrylonitrile-Butadiene-Styrene
PA	-	Polyamide
PC	-	Polycarbonate
PP	-	Polypropylene
PS	-	Polystyrene
P/L	-	Parting line
HREG	-	Hot runner edge gating
TG	-	Tunnel gating
EG	-	Edge gating
h_r	-	Depth of recess
t	-	Thickness
D	-	Diameter
H	-	Height

LIST OF APPENDICES

APPENDIX	TITLE	PAGE
A	Drawing of cavity plate	74
B	Drawing of core plate	75
C	Drawing of locating ring	76
D	Drawing of sprue bush	77
E	Drawing of guide pin	78
F	Drawing of pin (ejector plate)	79
G	Drawing of spacer block	80
H	Drawing of ejector pin	81
I	Drawing of ejector plate	82

CHAPTER 1

INTRODUCTION

1.1 Project Background

Injection molding nowadays have been one of the most important industry in the world. By using this method, the production became faster and more productive. The developing of injection molding becomes a competition from day to day. This process now integrated with computer control make the production better in quality and better quantity.

In designing the mold for injection molding, the accuracy in making mold very important in order to reduce cost and also to make sure that the mold broke easily. Before this, the mold designer used manual analysis to the mold. But now, there is software that can simulate the analysis of the mold that wants to develop.

Clearly, more and more manufacturers are using computational and analytical techniques to reduced design time and cost while significantly improving yield and quality. By using plastics flow simulation products, the determination of manufacturability of part in the early design stages and avoids potential downstream problems which can lead to production delays and cost overruns.

Simulation software allows to do some trouble shooting very easily. Some of the materials that were use are very expensive. Therefore, less time on the production floor working through a problem saves labor and material costs. By using software, designers been able to run simulations and locate and eliminate unsightly nit lines.

Problems that can be avoided by performing flow analysis early in the design stages are air traps, warpage, sink marks and voids, shrinkage, weld lines and meld lines.

1.2 Problem Statement

The problems statement for this project is:

- i. Normally, more than three times mold testing required to balancing the material flow into the part especially for multi-cavities and family molds.
- ii. Improper gate size, gate location and runner size will caused the long time for mold testing to get the desired final appearance of the product.

1.3 Project Objective

The objective of the project is to analysis material flow for injection molding process in order to get the suitable runner and gate system and design the two-plate injection mold for UMP keychain.

1.4 Project Scope

In order to achieve the objective, the project scopes must include:

- i. Literature review on injection molding mold.
- ii. Design the UMP keychain using Solid works.
- iii. Analysis the material flow using Mould flow software in order to select the suitable runner and gate system.
- iv. Design the two-plate injection mold for UMP keychain.
- v. Injection mold die is from standard die

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The best way is the molds and parts designer must have a good knowledge on the basic of injection molding process. It will help them to design for manufacturer and not just a design which is very nice but can't be manufactured. In this chapter, we will look at the basic theory of plastic injection molding including type of material, product design, mold design and process.

2.2 Injection Molding

Injection Molding is the process of forcing melted plastic in to a mold cavity. Injection molding is used for processing thermoplastics, thermosets, and elastomers. This is a high-rate production process and permits good dimensional control. Injection molding is a versatile process capable of producing complex shapes with good dimensional accuracy and at high production rate. [5]

The advantages of injection molding are high production rates, high tolerances are repeatable, wide range of material can be used, low labor costs, minimal scrap losses, little need to finish parts after molding and design flexibility.

2.2.1 Raw Materials

Most raw materials can be used. The resin is in pellets before processing. The examples of the material are Acrylonitrile-Butadiene-Styrene (ABS), Polyamide or Nylon (PA), Polycarbonate (PC), Polypropylene (PP), and Polystyrene (PS).[3]

2.2.2 Injection Molding Cycle and Process

The basic injection cycle is as follows: Mold close - injection carriage forward - injects plastic - metering - carriage retracts - mold open - eject part.

The molds are closed shut, by hydraulics or electric, and the heated plastic is forced by the pressure of the injection screw to take the shape of the mold. Some machines are run by electric motors instead of hydraulics or a combination of both. The water-cooling channels then assist in cooling the mold and the heated plastic solidifies into the part. Improper cooling can result in distorted molding or one that is burnt. The cycle is completed when the mold opens and the part is ejected with the assistance of ejector pins within the mold.

The resin, or raw material for injection molding, is usually in pellet or granule form, and is melted by heat and shearing forces shortly before being injected into the mold. Resin pellets are poured into the feed hopper, a large open bottomed container, which feeds the granules down to the screw. The screw is rotated by a motor, feeding pellets up the screw's grooves. The depth of the screw flights decreases towards the end of the screw nearest the mold, compressing the heated plastic. As the screw rotates, the pellets are moved forward in the screw and they undergo extreme pressure and friction which generates most of the heat needed to melt the pellets. Heaters on either side of the screw assist in the heating and temperature control during the melting process.

The channels through which the plastic flows toward the chamber will also solidify, forming an attached frame. This frame is composed of the sprue, which is the main channel from the reservoir of molten resin, parallel with the direction of

issues related to the wall but also impact to on manufacturability such as injection pressure required fill, cooling time and influence on ejection from the mold. [2]

Maintaining a constant wall thickness should be primary objective of product design. Each region in part has difference thickness will tend to shrink differently. This variation shrinkage will not only complicate achieving the desired size of part but also major contributors to warpage and residual stress. If variation in wall thickness cannot be avoid then try to keep the variation to minimum and provide a gradual transition rather than sudden change (see figure 2.1). [2]

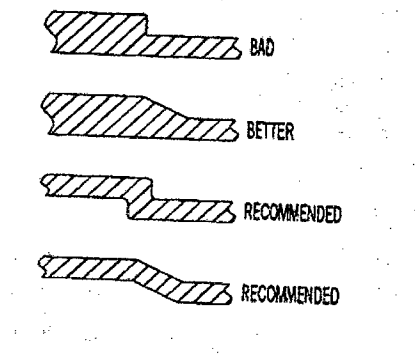


Figure 2.1: Recommended change at the intersection of thick/thin region [2]

2.3.2 Corner, Fillet and Radii.

Figure 2.2 illustrated the proper design of corners in a plastic part. This design provides for a constant wall thickness throughout the corner region. The inside radius should be a minimum of 0.5 (50%) the thickness of primary wall. The outside radius should be the inside radius plus the wall thickness. This ensures that the wall thickness is constants throughout the corner. (Note: the larger the inside radius is better) [2]

draw, and runners, which are perpendicular to the direction of draw, and are used to convey molten resin to the gate, or point of injection. The sprue and runner system can be cut or twisted off and recycled, sometimes being granulated next to the mold machine. Some molds are designed so that the part is automatically stripped through action of the mold. [10]

2.3 Part Design Guideline for Injection Molded Plastic Part

During part designing stage, the designers must consider whether the process has been choice has a capability to meets the design requirement such as size of part, shape, cosmetic appearance and tolerance.

The basic factors that should be considered during part designing are:

- i. Use uniform wall thicknesses throughout the part. This will minimize sinking, warping, residual stresses, and improve mold fill and cycle times.
- ii. Use generous radius at all corners. The inside corner radius should be a minimum of one material thickness.
- iii. Use the least thickness compliant with the process, material, or product design requirements. Using the least wall thickness for the process ensures rapid cooling, short cycle times, and minimum shot weight. All these result in the least possible parts cost.
- iv. Design parts to facilitate easy withdrawal from the mold by providing draft (taper) in the direction of mold opening or closing.

2.3.1 Wall Thickness

Plastic parts are generally designed around the use of relatively thin walls. Unless the injection molded plastic part is produced with gas assist injection molding, its walls are normally less than 5mm. When determining the thickness of the primary wall one must consider not only the structural, functional and aesthetic

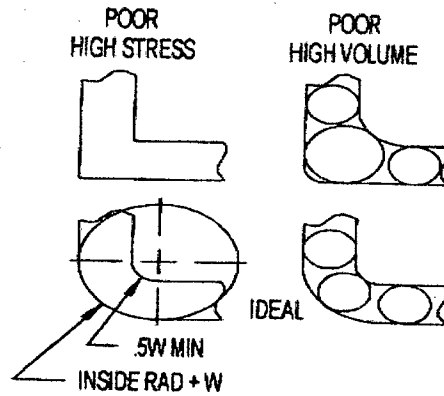


Figure 2.2: Proper designs of corners in a plastic part [2]

2.3.3 Rib

Ribs should follow the proportional thickness guidelines shown in figure 2.3. If the rib is too thick in relation to the part wall, you may experience sinks, voids, warpage, weld lines, and longer cycle times. Position ribs in the line of flow to improve filling and prevent air entrapment [8]

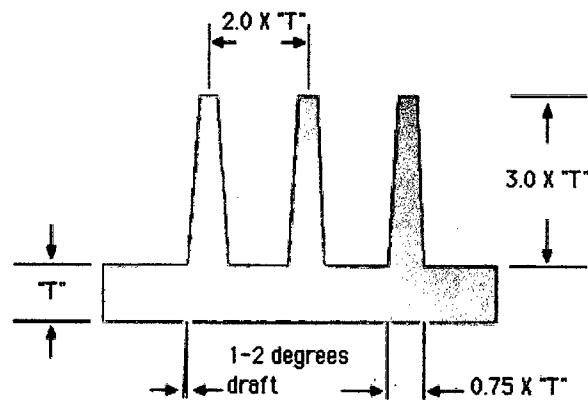


Figure 2.3: Guideline for design of rib respect to wall thickness [8]

2.6.1 Bosses and Gussets.

Bosses are used in parts that will be assembled. Connect the boss to a wall or rib with a connecting rib as shown in Figure 2.4. If the distance of the boss from the wall makes a connecting rib impractical, design the boss with gussets as shown in Figure 2.5.[8]

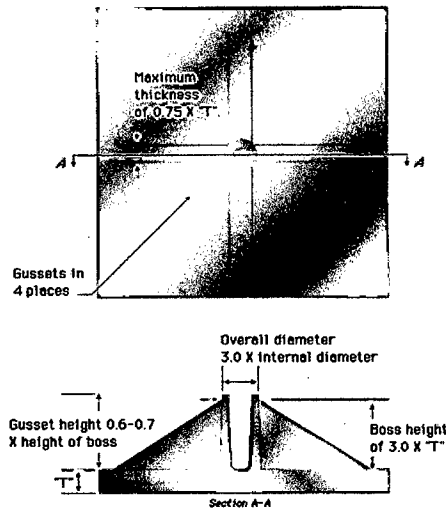


Figure 2.4: Free standing boss with four gussets. [8]

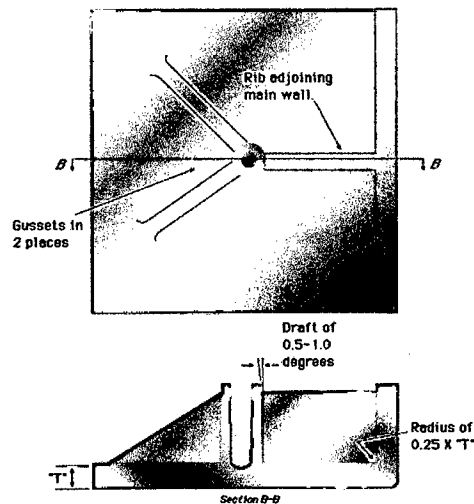


Figure 2.5: Free standing boss with two gussets [8]

2.3.5 Taper and Draft Angles

Draft or tapers are angles put on vertical walls of injection molded part to provide easier ejection from the mold. These angles generally range $1/8^\circ$ to several degrees depending on material, anticipated ejection problems, and product design requirements. Draft on inside and outside surfaces of side walls should be equal and parallel (Refer figure 2.6). The greater of draft, less potential for ejection problems. [2]

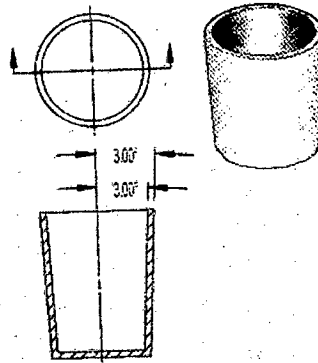


Figure 2.6: Draft on the inside and outside surface of part should be equal and parallel. [2]

2.4 Injection Mold Design

An injection mold must satisfy the following basic requirements:

- i. Contain a core and cavity set that defines the features of part will form.
- ii. Provide means for molten plastic to be delivered from the injection molding machine to the part forming cavities.
- iii. Act as heat exchanger, which will cool the part rapidly and cool the part uniformly
- iv. Provide for the molded part to be ejected from mold
- v. Have a structure that will resist internal melt pressure.

- vi. In multicavity molds, provide uniformly to each cavity through dimension, melt delivery, and cooling

All of the above is done with mold steel tolerances often as little as ± 0.00508 mm (0.0002 in). [1]

2.4.1 Type of Injection Mold

There are three basic types of molds:

- i. The cold-runner two plate mold (design is simplest)
- ii. The cold-runner three-plate mold, in which the runner system is separated from the part when the mold opens.
- iii. The hot-runner mold (also called runnerless mold), in which the molten plastic is kept hot in a heated runner plate.

In cold-runner molds, the solidified plastic in the channels that connect the mold cavity to the end of the barrel must be removed, usually by trimming. This scrap can be chopped and recycled. In hot-runner molds, which are more expensive, there are no gates, runners, or sprue attached to the molded part. Cycle times are shorter, because only injection molded part must be cooled and ejected. [6]

2.4.2 Standard Mold Assembly

Basic type mold called “two plate molds”. When the mold opens, the mold is divided into two blocks of fixed side (cavity side) and the movable side (core side). The ejector mechanism is provided on the movable side.

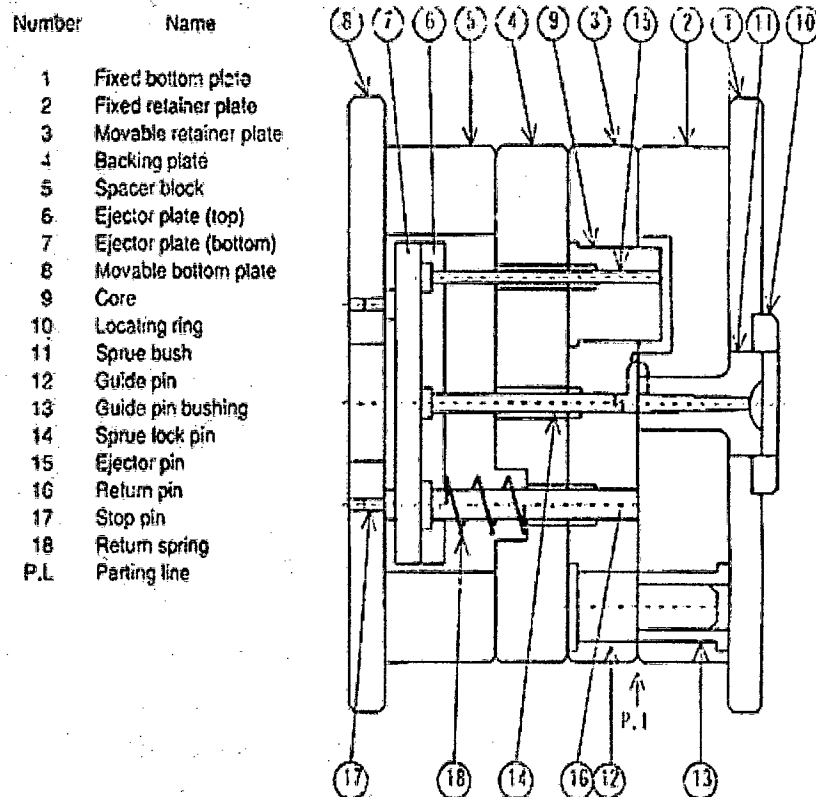


Figure 2.7: Standard mold [7]

2.4.2.1 Mold Part Name and Functions

Explanations are given here on names and function of mold parts based on standard mold show in figure 2.7.

The right side from the boundary surface of mold in opening, i.e. the parting line (P.L), is called the “fixed side”, and the left side the “movable side”. The fixed side means that the parts are fixed to the nozzle side of the molding machine. The movable side means that parts are mounted on the movable platen sliding for opening/closing of mold and move per injection shot.

The locating ring (10) on the fixed side is fitted on a hold of same diameter in the bottom plate of a molding machine for location of a mold when it is mounted on the molding machine.