ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Hot Runner Mould Design and Plastic Flow Analysis for CAP


V.Chandra Sekhar
M.Tech (CAD/CAM),
Department of Mechanical Engineering,
AITS, Rajampet, Andhra Pradesh, India.

N. Jaya Krishana
Assistant Professor,
Department of Mechanical Engineering,
AITS, Rajampet, Andhra Pradesh, India.

ABSTRACT:

Injection moulds are divided in to two types based on runner design (i.e) Cold runner moulds and Runner less moulds (i.e) hot runner moulds. In cold runner moulds, for multi-cavity and multi point injection moulds, there is wastage of material in runner area. Sometimes wastage of material is more than component weight. For avoiding the above problem, the technique used is hot Runner moulds. Hot runner mould is one of the advanced manufacturing methods for multi cavity type moulds. These types of moulds are commonly used for large production rate.While producing plastic components using normal/standard multy cavity mould, we are facing the problems like partial filling, cavities in components, injection pressure and temperature drop age etc... In this project, those problems are rectified by using hot Runner moulds.

I.INTRODUCTION:
Hot- runner unit moulds:

The Moulds which contains a heated runner manifold block within its structure are named as hot-runner unit moulds. The block suitably insulated from the rest the mould, is maintained at a closely controlled elevated temperature to keep the runner permanently as a melt. The polymer material can thereby be directed to mould extremities without loss of heat and without pressure loss associated with temperature variations.

Advantages:

- There is no feed system for the operator to remove from the mould.
- On manually controlled injection machines the mould open time is reduced.
- The cost of separating strong and regrinding scrap feed systems is thereby saved.
- As no scrap material is produced it cannot be contaminated.
- The moulding is automatically degated on ‘direct feed’ design.

Limitations:

- The mould setting time is generally greater than for a corresponding two-part mould.
- An extended period, waiting for the hot-runner unit to heat, is required before production can commence.
- The initial ‘debugging’ of a new hot runner mould is usually more extensive than with a standard mould.
- The cost is higher than that of a standard mould and in some cases than that of an underfeed (three-part) mould.

Applications:

- It allows for the pin gating of mouldings on multi-impression and multi-of mould
- It allows for multi-point gating on single-impression and multi-impression moulds.
- It allows for side or film gating of large moulding
- It permits the semi-runner less design to be adopted where small groups of impressions are fed from secondary sprues.

II.INJECTION MOULDING:
Injection moulding is a manufacturing process for producing parts from both thermoplastic and thermosetting plastic materials.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Material is fed into a heated barrel, mixed, and forced into a mold cavity where it cools and hardens to the configuration of the mold cavity. After a product is designed, usually by an industrial designer or an engineer, molds are made by a moldmaker (or toolmaker) from metal, usually either steel or aluminum, and precision-machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars.

Resins are molded throughout the world in a wide variety of types and designs of injection and extrusion equipment. The first purpose of the injection unit for molding crystalline materials to deliver to the mold the necessary amount of a homogeneous melt (with no un-melt and no degraded material).The rules of construction of the injection unit are then dependent on the molding material requirements in term of thermal behavior and heat needed .

The first point to take into account for a crystalline material is the thermal stabilityat melt temperature, to avoid degradation. Then, screw, nozzle, backflow valve, adaptor, should be designed to provide efficient melting of crystalline material and delivery of molten polymer to the mold .Two rough methods to evaluate the presence of un-melt and of degraded material will be presented in "Evaluation of Melt Quality".

Thermal Stability during Processing:

The amorphous polymer starts softening just after Tg and presents a continuous change in viscosity. This gives a very large temperature range to operate (but a large variation of viscosity with temperature). In contrast, the crystalline polymer stays solid up to the melting point and suddenly melts to the liquid phase at high temperature. This limits the processing range of temperature between un-melt and thermal degradation (specifically for Delrin®190–250°C [374–482°F]) .The second factor is the time the material stays at that temperature. For all polymers, the molecules can withstand a certain time at a certain temperature before degradation can start. Obviously this acceptable time limit becomes shorter when the temperature is higher. Degradation will result in generation of gases which cause bubbles in the melt, splays on parts, mold deposit, yellow and brown marks on the parts.

Average HUT =

$$\frac{weightofresignincylinder}{shotweight} \times \text{cycle time}$$

A quick approximation can be done by:

Average HUT =

$$\frac{maximunscrewstroke \times 2}{currentscrewstroke} \times \text{cycle time}$$

With a screw stroke of 1 diameter (a small shot) and a cycle time of 1 min (a very long one), the average HUT is equal to 8 min. At the recommended melttemperature of 215°C (419°F), the maximum HUT is over 30 min.

Screw Design:

Screw design is a key parameter for productivity, because for crystalline materials the screw rotation time is an inherent part of the cycle time.As mentioned above, it should take in consideration the specific melting behavior of the crystalline material, i.e., solid up to the melting point, high demand of heat during melting and low viscosity of the molten material.Although general-purpose screws are widely used for molding exceeding the output capability of an inadequately designed screw will cause wide temperature variations and unmelted particles (sometimes unmelt and degraded material have been observed at the same time). The result is loss of toughness, variability in shrinkage and dimensions, warping, surface defects, plugged gates (leading to short shots) or other molding problems.

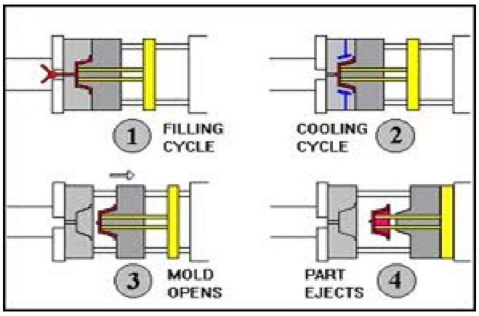


Figure 1: Injection process

III.MOLDS:

A mold or mould is a hollowed-out block that is filled with a liquid like plastic, glass, metal, or ceramic raw materials.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

The liquid hardens or sets inside the mold, adopting its shape. A mold is the counterpart to a cast. The manufacturer who makes the molds is called the mold maker. A release agent is typically used to make removal of the hardened/set substance from the mold easier. Typical uses for molded plastics include molded furniture, molded household goods, molded cases, and structural materials.

Mould design:

The mold consists of two primary components, the injection mold (A plate) and the ejector mold (B plate). Plastic resin enters the mold through a sprue in the injection mold, the sprue bushing is to seal tightly against the nozzle of the injection barrel of the molding machine and to allow molten plastic to flow from the barrel into the mold, also known as cavity .

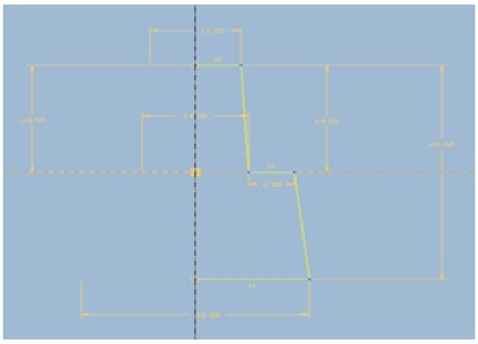
The sprue bushing directs the molten plastic to the cavity images through channels that are machined into the faces of the A and B plates. These channels allow plastic to run along them, so they are referred to as runners. The molten plastic flows through the runner and enters one or more specialized gates and into the cavity geometry to form the desired part.

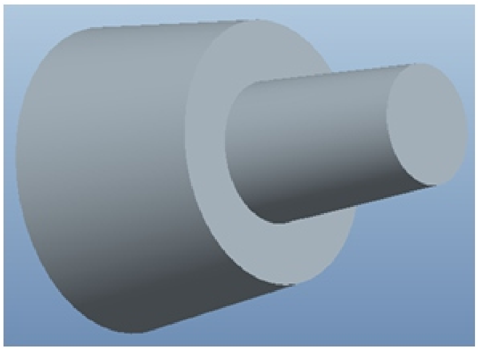
Types of moulds:

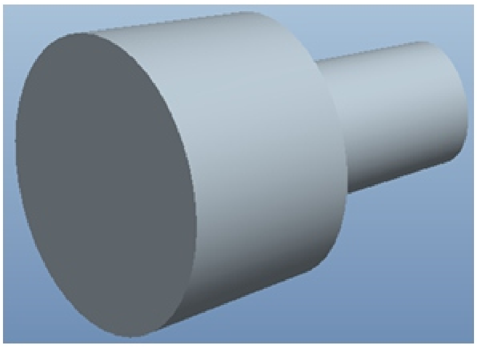
While there are literally hundreds of different types of molds, Matrix deals primarily in thermoplastic injection molds. Our tools are designed and built to run in presses of 500 tons or smaller. We are particularly well-suited to complex, miniature and micro-tooling projects.

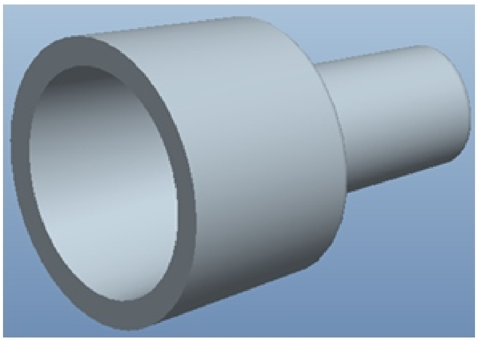
- Prototypes
- Low / High-Cavitation
- Family Mold
- Unscrewing Molds
- Multiple Shot
- Hot Runner
- Cold Runner
- Insulated Runner
- Two / Three Plate
- Unit Die
- Micromold

IV.MODEL OF CAP:









ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

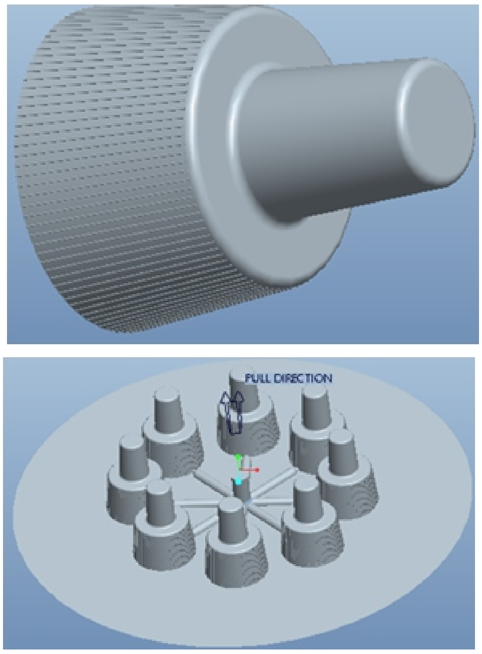


Figure 2: Design of CAP in PROE.

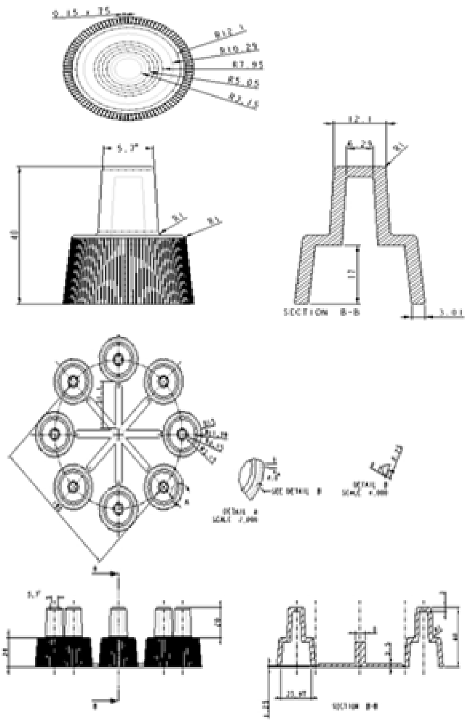


Figure 3: Drafting's.

V.MOULD FLOW ANALYSIS:

Mould flow, 3D solids-based plastics flow simulation that allows plastics part designers to determine the manufacturability of their parts during the preliminary design stages and avoid potential downstream problems, which can lead to delays and cost overruns. Following are the benefits:

• Optimize the part wall thickness to achieve uniform filling patterns, minimum cycle time and lowest part cost Identify and eliminate cosmetic issues such as sink marks, weld lines and air traps.

• Determine the best injection locations for a given part design Mould flow analysis gives you the ability to maintain the integrity of your product designs. It provides you the tools to quickly optimize part designs and check the impact of critical design decisions on the manufacturability and quality of the product early in the design process.

There is no need to:

• Compromise the aesthetics of your design concept for manufacturability;

• Go through a lengthy trial and error process to find the most suitable material to produce the part with the highest possible quality and the lowest possible cost

• Find out during trial runs that the produced part has visual blemishes, such as sink marks, weld lines, air traps or burn marks.

Plastic advisor in pro/engineer:

Problems found after tooling development are always expensive and frustrating. For plastic part design and manufacture, there is a better way. By simulating the plastic-filling process for injection-molded parts, Pro/ENGINEER Plastic Advisor enables engineers to design for manufacturability, uncover problems, and propose remedies, reducing development time and expense. Pro/ENGINEER Plastic Advisor simulates mold filling for injection molded plastic parts. Advanced features provide valuable manufacturability insight - insight that can significantly reduce late-cycle design changes and mold reengineering costs.

Features & Benefits:

• Animates plastic injection fill process and automatically creates Web reports within Pro/ENGINEER browser.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

- Access library of common plastic materials and automatically select from typical injection-molding machine parameters

- Identify optimal injection locations to reduce cycle time and improve product appearance

- Identify potential mold-filling problems such as short shots, air traps, and weld lines

- Improve design quality and reduces manufacturing cycle times and rework of molds.

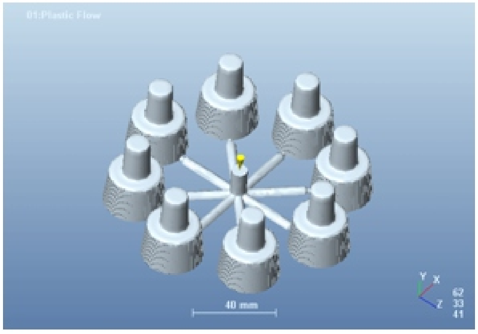


Figure 4: Plastic flow analysis

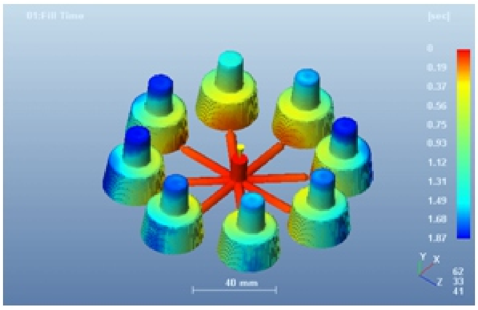


Figure 5: Fill time

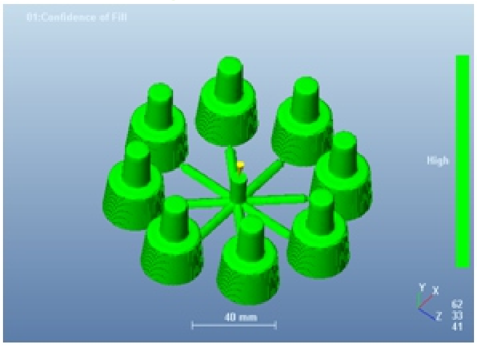


Figure 6: confidence of fill

VI.MOULD CALCULATIONS:

Clamping Tonnage

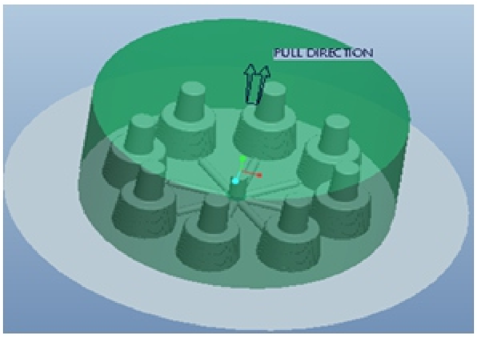
Fc = Pc×Ap×n
Fc = clamping tonnage. (Tons)
Pc = Cavity pressure =101.9 Kg/ cm2
Total pressure=101.9×9=917.4 Kg/ cm2
Ap = projected area = 56.547 cm2
n = no. of cavity =9
Fc = 917.4×56.547×9 = 466.85 tons
Considering factor of safety 466.85×1.2=560tons
Available tonnage = 850
As per machine standards = 850 tons
= 850×0.85 = 722.5 tons
J850AD (JAD series)
Mold height = 500 to 1100 mm
Plate size=1590×1590
Injection pressure=185mpa
Injection speed=160mm/sec
Heater wattage=44.5kw
Mechanism-double toggle
Clamping force-850tons
Maximum daylight opening=2300
Based on the shot capacity
Shot weight = 504×density of PA-6/density of PS
= 504× (1.45/1.05) = 696 gm
Considering the Factor of safety 85% only
= 696×0.85 = 591.6

Cooling calculations

Q = Heat to be transferred per hour by plastic Material= Mp×a Cal /hr
Mp = Mass of plastic Material injected into mold per hour in gms/hr
a = heat content of plastic in Cal/gm = 50
Mp = Shot weight × no.of cycles per/hr
No.of cycles per /hr
56.6/comp filling time = 60 sec
3600sec/hr/90 per/comp =60 comp
Mp = 696×60 = 41760
Q = 41760×5 = 208800 Cal/hr
Qw = Rate of heat to be extracted by water in Cal/hr
K = the constant to allow heat transfer efficiency
Mw = Mass of water circulated in gms / hr
Qw = Mw × K(tout –tin) 50
For direct cooling method = 0.64
Indirect method = 0.50
Qw = Mw × K(tout –tin) = Q/2
Mw = 208800/2×0.64×5 = 334.08 lit/hr

Core and cavity extraction

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal



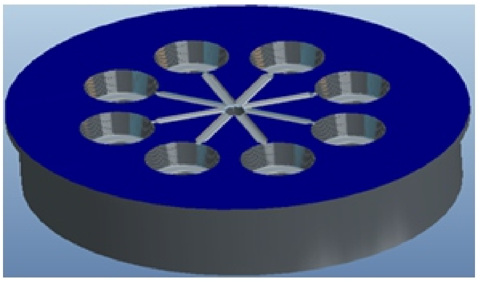


Figure 7: Cavity

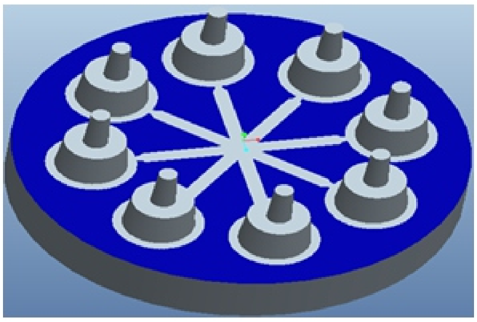
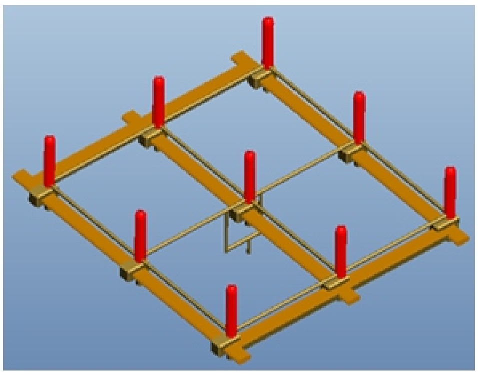


Figure 8: CORE



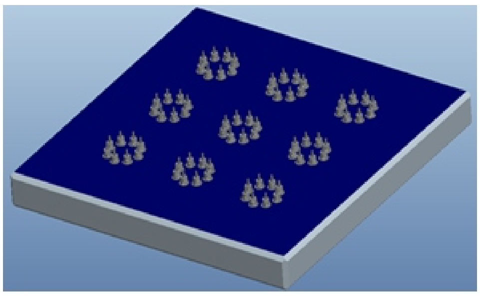


Figure 9: Entire mold assembly

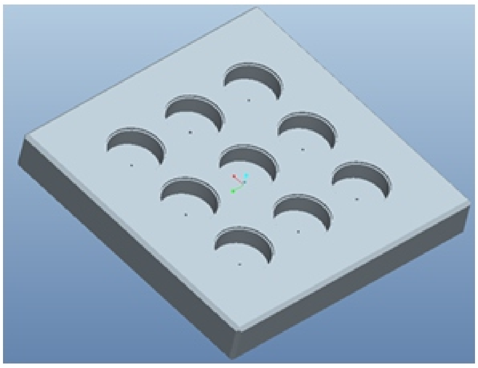


Figure 10: The core insert of the mold

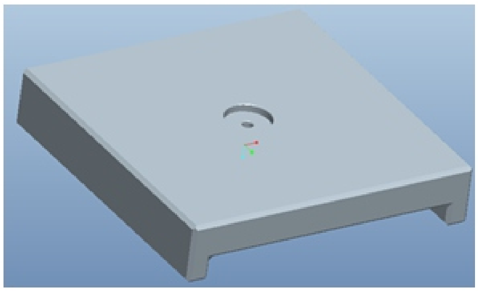


Figure 11: the cavity back plate of the mold

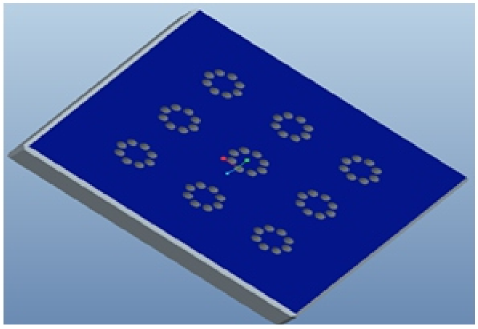
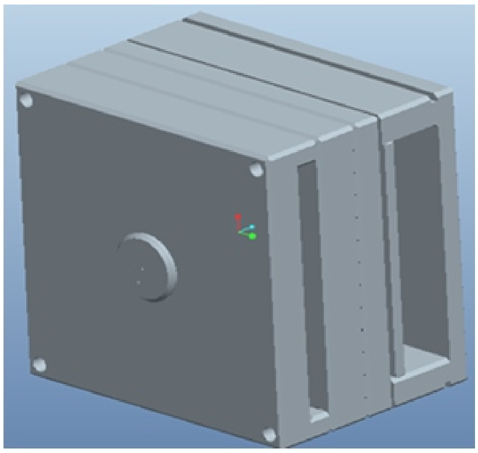


Figure 12: the core back plate of the mold



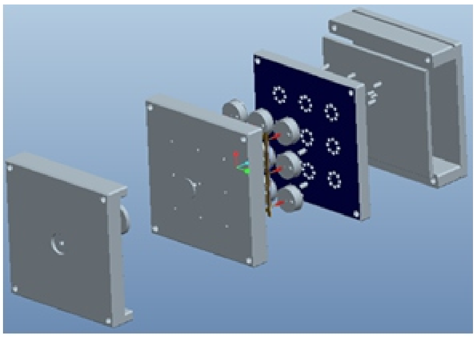


Figure 13: Entire set up of hot runner mold

VII.CONCLUSION:

This paper deals with complete hot runner mould tool design as per the parameters provided by client.Core and Cavity is extracted for the cap pattern with the usage of parting surface in mfg segment of pro/engineer. Mould Flow Analysis is done on cap set which is made for the cavity patterned components.Usage of mould flow analysis for finding the material filling, pressure distribution, air traps, weld lines formed during injection molding process.By simulating the plastic-filling process for injection-molded parts, Pro/ENGINEER Plastic Advisor enables engineers to design for manufacturability, uncover problems, and propose remedies, reducing development time and expense. By using hot runner mould we can increase the production rate of caps 10 times than general mould.Using hot runner mould, reduction of time in production so that we can increase the production rate.By using this analysis and manufacture process cap mould design can be done without any failures.

REFERENCES:

1)Design and manufacturing of air conditioner panel hot runner injection mould based on CAD/CAE/CAM by"WUJI JIANG" and "ZHIGANG JIN".

2)The applied study on hot runner technology and design of injection moulds by "HUANG and YANG".

3)Machine design, R.S.Khurmi/J.K.Guptha (S.CHAND)
4)Design data book: P.S.G.College of Technology (Kalaikathirachchagam),

5)Drag material change in hot runner injection moulding by "LEE".

6)Design of machine element: V.B.Bandari (TATA Mc Graw-hill

7)Injection mould design: R.G.W. PYE (East-West press Pvt. Ltd