Prakash Kumar Vishwakarma, 10:1
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

High Temperature Pattern and Mould Analysis

Prakash Kumar Vishwakarma, Prof. G.R. Kesheorey
Department of Mechanical Engineering,
Vindhya Institute of Technology & Science,
Indore M.P.

Abstract- Plastic moulding is the process of shaping plastic using a rigid frame or mould. The technique allows for the creation of objects of all shapes and sizes with huge design flexibility for both simple and highly complex designs. A popular manufacturing option, plastic moulding techniques are responsible for many car parts contender sign and other high volume items.

Keywords- Plastic moulding, high volume etc.

I. INTRODUCTION

The history of rotational moulding began some were between 1940 and 1950 in the USA when the presses was developed for a small number of plastics. By the late 1950s, when the rotational moulding process was better understood, Applications for other industry were developed including road corms marine buoys, armrests. A process was developed in Europe in the early 1960s that enabled large hollow containers to be created in Low Density Polythylele [LDPE] by rotating [or rocking] a mould on a chassis, houses open gas jets, through 30 degrees which coated the inside of the mould with the polymer.

The history plastic injection moulding has seen steady industry growth since its beginnings in the late 1800's the technique has evolved from the production of combs and buttons to major consumer, industrial, medical and aerospace products. In 1868, John Wesley Hyatt invented a way to make billiard balls by injecting celluloid into a mould. [1]

The blow moulding process, in comparison with injection moulding, is a low pressure process with typical blow air pressures around 25 to 150 psi.

There are several types of blow moulding process. Injection blow moulding is used in the production of large quantities of hollow plastic objects. Typically used to make small medical and single serving bottles, injection blow moulding is the least-used of all blow moulding processes. Extrusion blow moulding process begins with conventional downward extrusion of tube.

When the tube reaches the desired length the mould is closed catching and holding the neck end open and pinching the bottom end closed. Then a blow pin is inserted into neck end of hot tube to form the threaded opening and inflate the tube inside the mold cavity.

When the mould is completely cooled it is opened to eject the bottle and the excess plastic is trimmed from the neck and bottom areas. Stretch blow moulding includes jars, bottles and similar container because it produces items of excellent visual and dimensional quality compared to extrusion moulding.

Compression moulding is a high volume, high pressure plastic moulding method that is suitable for moulding complex, high strength object and with its short cycle time and high production rate, many organization in the automotive industry have chosen compression moulding to produce parts.

The manufacturing industry for plastic product is growing rapidly in recent years, and more and more plastics are used widely to substitute for metals. Injection moulding is the most popular thermo plastic moulding process having several advantages such as short product cycle, excellent surfaces of product and easily surfaces of product and easily moulded complicated shapes. Generally it consists of three phases, filling, packing and cooling.

The filling stage is the critical stage in the production of good quality moulding. In the filling stage, injection moulding process parameter include melt flow rate, injection pressure, mould temperature and

melt temperature, some of which have a direct influence on the quality of the injection products.[2]
It is a human tendency to develop the new things according to our need. To modify innovative ideas which is our main goal to develop systems and components that are characterized through their reliability and user-friendliness?

The results of our developments are optimal balanced and custom build Hot Runner Manifolds as well as a modular system of reliable Hot Runner Nozzles for almost all injection molding applications. As important it is to choose a new technology that offers high-quality products for reasonable prices and ensures on time delivery – in today's world it is more than ever important to choose a partner who develops and delivers optimal solutions based on extensive knowledge and dedication.

Basis of our work are always the specific application needs of our humanity. Whether supporting the selection of Hot Runner Nozzles for a new project, providing assistance in determining the ideal gate design or the integration of our Hot Runner Systems within the mold – our experienced specialists are always available with their extensive knowledge and are satisfied only when the optimal solution is determined.

There is a need to design, develop and prepare drawings for hot runner system for varied and challenging applications including home ware, containers, automotive, engineering products and etc. Prepare and enhance drawing requirements to ensure that system requirements are based on customer's given specifications.

Provide technical support in modification of mature tooling for production. To assist in all activities related to hot runner or mould designs and modification. To avoid unnecessary molding costs, specify tight tolerances only when needed.

Generally, the size and variability of other part features determine the actual tolerance required for any one component or feature within an assembly. Rather than dividing the allowable variability equally over the various features that govern fit and function, allot a greater portion of the total tolerance range to features that are difficult to control and reserves tight tolerances for features that can accommodate them reasonably.

II. EXPERIMENTAL SET UP AND ANALYTICAL CALCULATION

Air cooler tank can be manufactured as per dimension by designing the mould tool with parameter for air cooler tank. The manufacturing process of air cooler tank is shown in the flow chart given below.

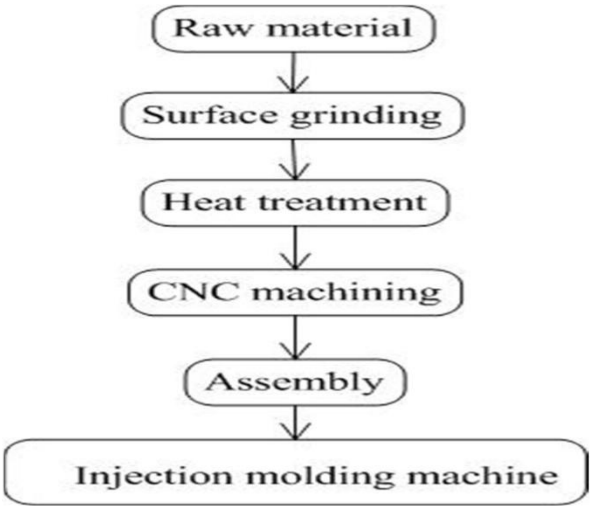


Fig 1. Flow Chart.

1. Raw Material:
Generally used mould tool materials are hot die steels. They posses excellent toughness, ductility and harden ability. Suitable for large dies having thickness greater than 200mm. Intricate dies and dummy block liners can be manufactured by using hot & warm forging and extrusion tooling.

2. Surface Grinding:
Surface grinding is performed after selection of raw material. Surface grinding is extensively used machining process in which chips of metallic and non-metallic substance are removed through a spinning wheel covered with rough particle.

3. Heat Treatment:
The material is heat treated to improve the strength of material. Heat treatment plays a vital role while manufacturing machine parts and tools. To alter the physical and mechanical properties without changing the shape of product on controlled heating and cooling of metal is known as Heat treatment. Heat treatment is often related to improve the strength of material, but it can also be used to change the certain objective of manufacturability such as improving machinability, formability and

restore ductility after cold working operation. Hence heat treatment is a skilled manufacturing process which not only helps other manufacturing process, but also boosts the performance of product by increasing strength or other necessary characteristics.

4. CNC Machine:

While using CAD/CAM programs in manufacturing process such as CNC, end-to-end component design is highly automated. A CAD/CAM program is having a set of computerized set of information or file which is needed to operate a particular machine by extracting the commands and then loaded into the CNC machines for production.

Since for manufacturing any particular component it might use number of different tools - drills, saws, single point cutting tool etc. – in modern machines often multiple tools are combined in single unit called "bit or cell". In transformation cases a different quantity of machines are used with an external controller and a robot or human operator so that the component move from machine to machine.

In either case to manufacture a highly automated parts which closely matches the original CAD design the complex series of steps needed. Authentication undergoing CNC machining process the mold tool i.e. core and cavity are shown in following figures.

5. Mould Calculation:

Clamping Tonnage

$$\mathbf{F_c = P_c \times A_p \times n}$$

F_c = clamping tonnage (tons)

P_c = Cavity pressure = 101.9Kg/cm^2

Total pressure = 101.9 × 9 = 917.4Kg/cm^2

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 toogle

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 transferred by plastic material per hour

Q = M_p × a Cal /hr

M_p = Mass of plastic Material filled inside mold per hour in gms/hr a = amount of heat in plastic in Cal/gm = 50

M_p = 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

Q = 208800 Cal/hr

Q_w = Amount of heat to be extracted by water in Cal/hr

K = the constant to allow heat transfer efficiency

M_w = Mass of water circulated in gms / hr

Q_w = Mw × K(tout –tin) 50

For direct cooling method = 0.64

Indirect method = 0.50

Q_w = M_w × K(tout –tin)

= Q/2

M_w = 208800/2×0.64×5

=104400×0.64×5

= 334080 gm/hr

= 334.08 lit/hr

III. SIMULATION AND MODELLING

1. Overview of Steps Involved in Solving the Problem.

Step 1: Data preparation

Step 2: Preparing drawing as per industrial standards

Step 3: Preparing pro-e models

Step 4: Mold flow analysis and comparison between normal and hot runner mold.

Step 5: Calculating mould parameters

Step 6: Preparing 3D models of general and hot runner molds

Step 7: Generating CNC codes
Step 8: Conclusion

2. Detailed Solving Procedure:

The following is the step-by-step procedure for the "Hot runner mould design and plastic flow analysis for cap" problem using the PRO-E & MOULD FLOW ANALYSIS in pro-e.

3. Building Geometric Model

Step-1: Dimension of cap are taken from available literature.

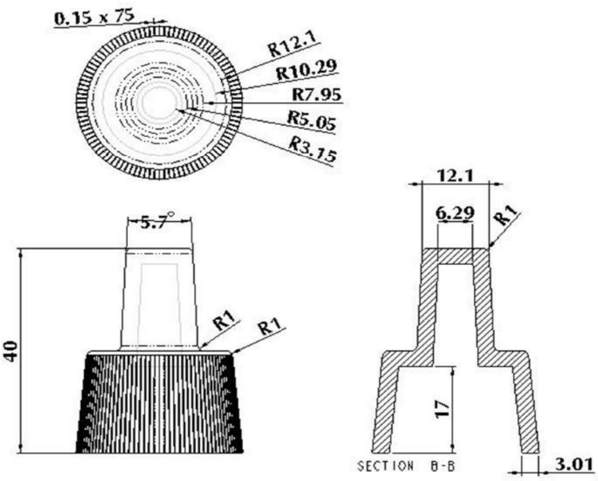


Fig 2. Drafting of Cap.

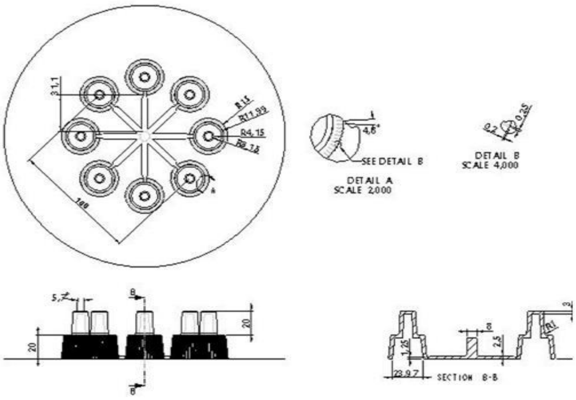


Fig 3. Drafting of Multi-cavity cap.

Step-2: Select Standard Dimension

- Select front plane
- Sketch
- Select "line" command l=3.14 horizontal, vertical 23 of angle 5.7 from horizontal plane than draw horizontal line 3mm again draw vertical line 17 mm at angle 5.7 from horizontal plane draw horizontal line to join central axis.
- Draw required profile using dimensions, thus model appears like below

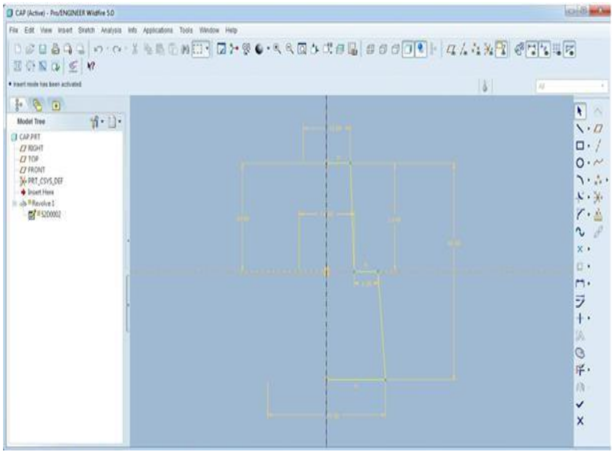


Fig 4. Sketch.

- Select revolve tool
- Specify center of axis for rotation
- Specify axis of rotation value of angle 360
- Select shell tool
- Select face to remove ie front plane
- Specify thickness value of 3.01mm
- Select round tool
- Select 3 edges
- Specify value of radius 1
- Select sweep tool
- Select trajectory ie bottom round of cap
- Specify section

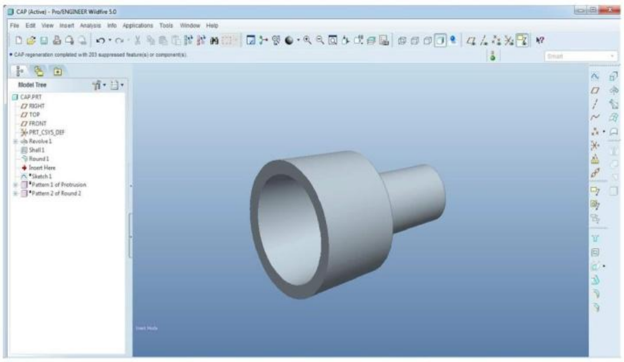


Fig 5. Part Design of Cap 17.Select pattern tool.

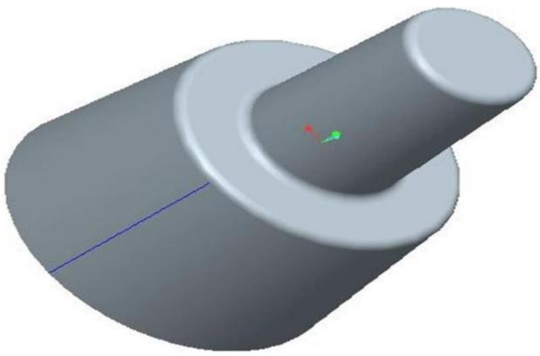


Fig 6. Actual Cap.

- Select axis pattern ie central axis
- Specify number of instances,
- Specify value=8

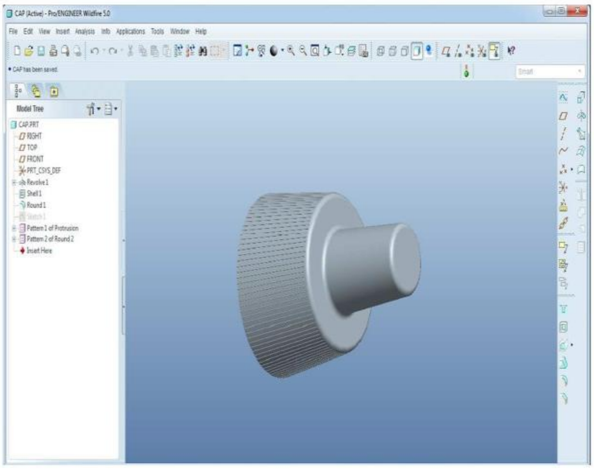


Fig 7. Cap in Pro-E.

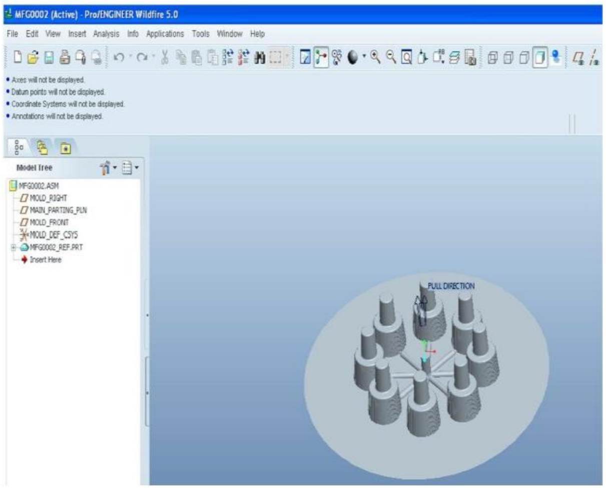


Fig 8. Cap solid model.

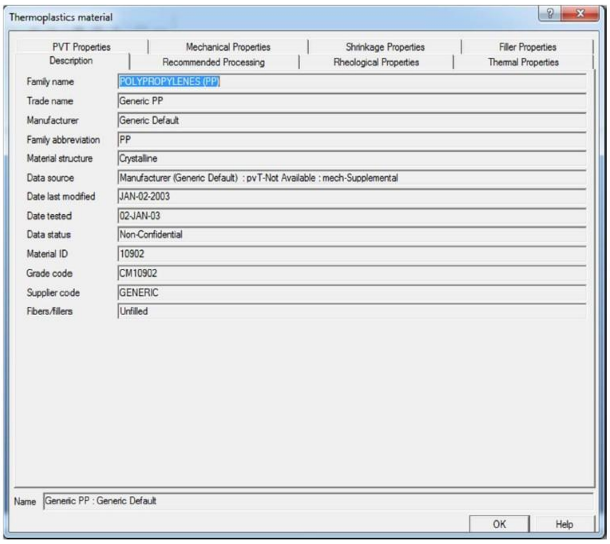


Fig 9. Thermoplastic material process 1.

To Do Analysis the Following Commands are Used:

- Select applications > plastic advisor >
- pick datum point for injection location > ok >
- select molding parameter icon >
- select specific material > select manufacturer
- pp.> select processing conditions
- enter melt temperature > enter mold temperature
- enter maximum injection pressure limit >
- Enter clamp time 9.ok.

IV. PLASTIC FLOW ANALYSIS

The Flow Analysis summary page gives an overview of the model's analysis, provides information about pressure, actual injection time and whether weld lines and air traps are present. In order to assess the mold ability of the part, the dialog uses the Confidence of Fill.

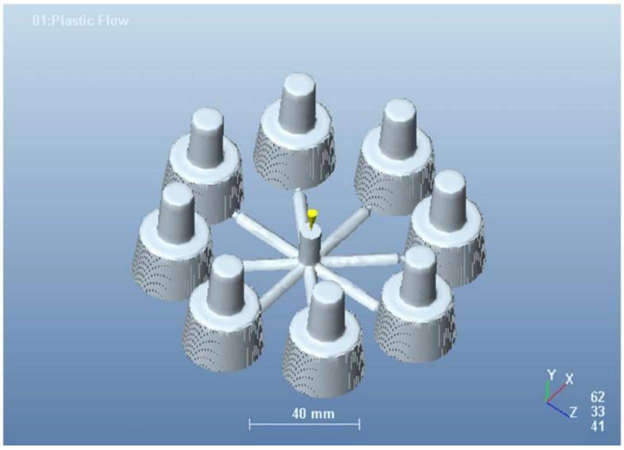


Fig 10. Plastic Flow in model.

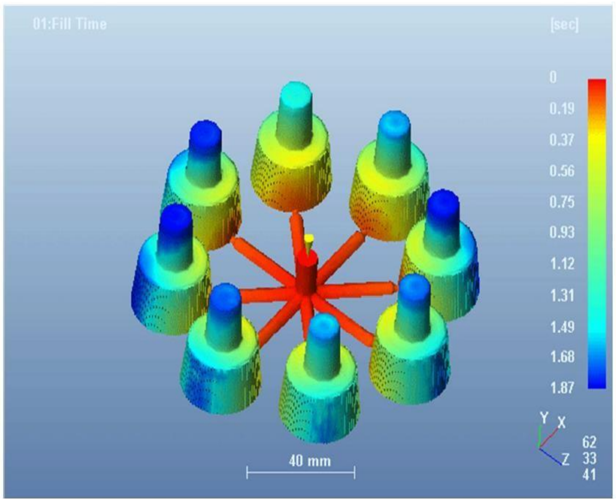


Fig 11. Plastic flow in fill time.

1. Fill Time:

This result shown by plotting contour of the part which joins regions of filling plastic at same time through the flow path of the plastic. These contours are displayed in a range of red color, to indicate the first region to fill, through to blue to indicate the last region to fill. A part of the model that did not fill is known as short shot, and will be displayed as translucent. The impression of plastic actually flowing into the mould is given by plotting these contours in time sequence.

2. Confidence of Fill:

The probability of a region within the cavity filling with plastic at conventional injection molding conditions is displayed as a result of confidence of fill. This result is derived from the pressure and temperature results.

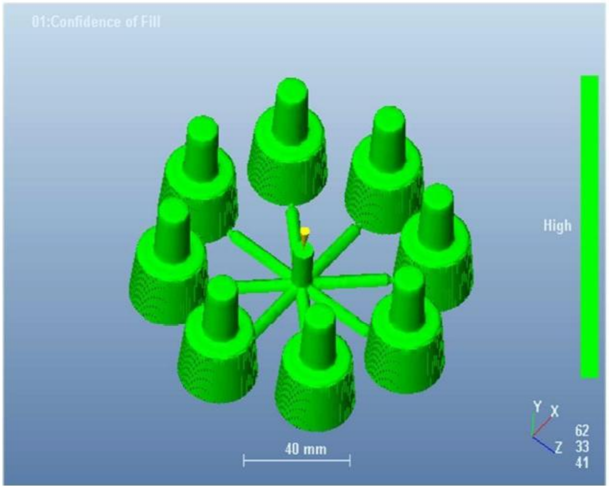
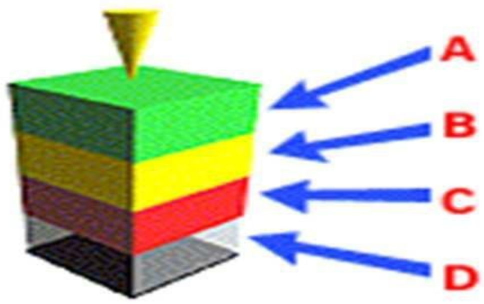


Fig 12. Plastic flow in confidence of fill.



- Will definitely fill
- May be difficult to fill or may have quality problem
- May be difficult to fill or may have quality problem
- Will not fill (short shot)

3. Pressure Result Derivation

There is always a pressure gradient from a maximum value at the injection location down to atmospheric pressure at the flow front at any point during Filling. The pressure distribution is measured continuously throughout the cavity filling by the adviser and presents you with 2 pressure results considered as a reference to the following simple part having single polymer injection location at one end:

The graph below shows how pressure varies over time at both the polymer injection location and at the point marked X.

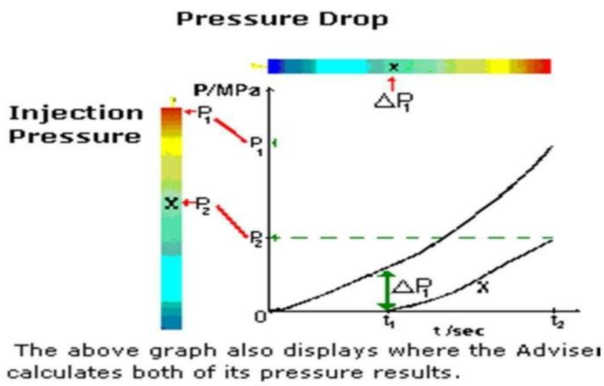


Fig 13. Graph.

- The Advisor calculates both of its pressure results in the above graph.
- The contour plot showing the pressure required to flow material to each point in the cavity is the result of Pressure Drop. The pressure at the injection location is a calculated value as a point (X in our example) fills and this value is plotted at the point with respect to X on the model. The drop in pressure is not displayed for any moment but the Injection Pressure result is displayed.
- The displayed value is related to the location in question (X) actually filled at that time.

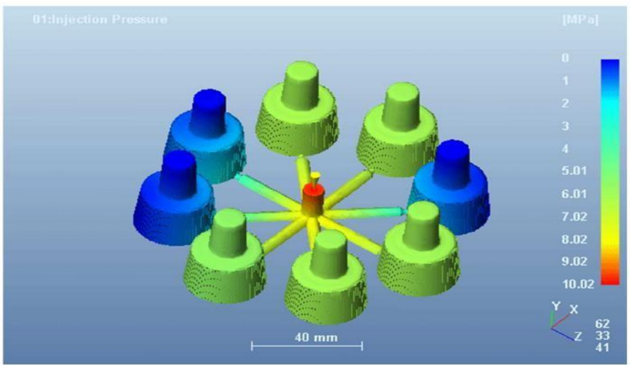


Fig 14. Injection pressure in model.

A contour plot of the pressure distribution throughout the cavity at the end of filling is the result of Injection Pressure. This effective image is taken at a instant of time. The above image shows that at the injection location value is maximum and minimum at the last point of the cavity to fill

4. Flow Front Temperature:

A range of colors is used to indicate the region of lowest temperature (colored blue) through to the region of highest temperature (colored red) shown in the result of flow front temperature. The color shows the material temperature at each point as that point was filled Changes in temperature of the flow front during filling are shown as a result.

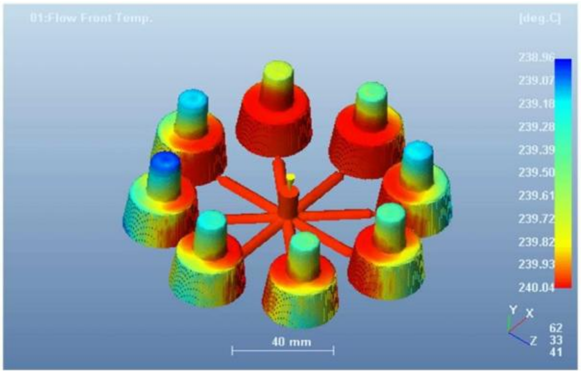


Fig 15. Flow Front temperature in model.

5. Quality Analysis:

The combination of the five results listed below derives the Quality display Analysis. These five results are classified as - unacceptable (red), acceptable (yellow) and preferred (green). The five results are:

- Flow front temperature
- Pressure drop
- Cooling time
- Shear rate
- Shear stress

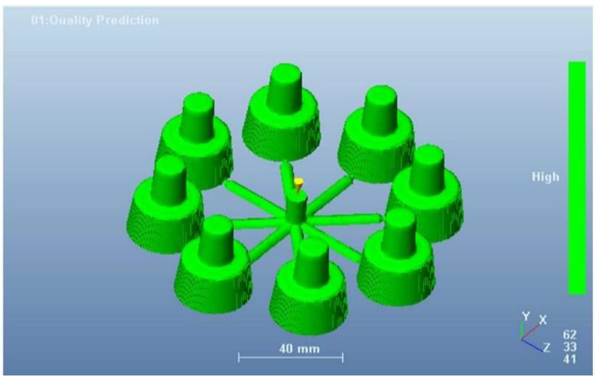


Fig 16. Quality Prediction in model.

6. Cooling Quality:

The entire time that is required to freeze all areas of the part completely and plots variations from that time value is calculated in the Cooling Quality analysis. The area need more cooling which has a greater freeze time than normal as indicated in a result to compensate for the heat that is concentrated in that area.

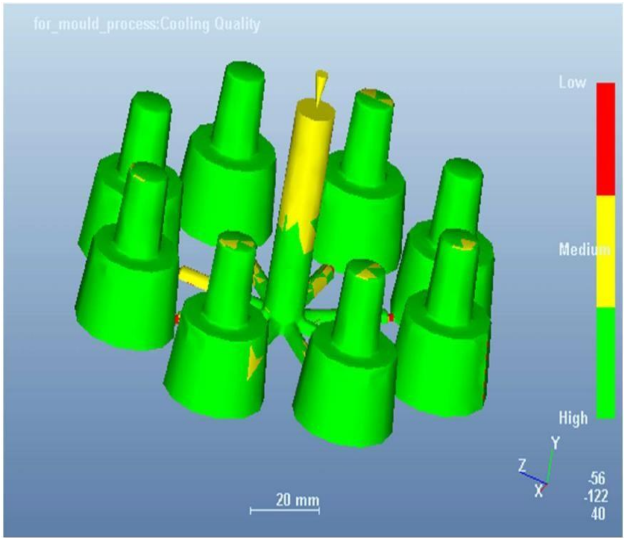


Fig 17. Cooling Quality in model.

7. The Surface Temperature Variance Is Higher Than Normal:

The surface temperature variance in a particular area of the model is higher than the average is warned by the Cooling Quality Analysis Adviser.

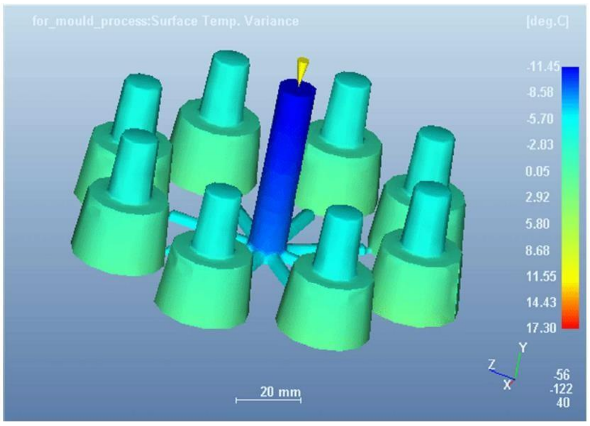


Fig 18. Surface Temperature Variance in model.

8. Sink Mark:

The location of Sink Marks (and voids) presence likely to be caused by features on the opposite face of the surface are indicated as a result. Sink Marks generally occur in moldings with thicker sections, or at locations opposite ribs, bosses or internal fillets.

The result does not clearly specify that locally thick regions are the main cause of Sink marks.

9. Important Information:

Users should undertake sufficient verification and testing to determine the suitability for their own particular purpose of any information presented herein. Nothing herein is to be taken as permission, inducement, or recommendation by Mould flow Corporation to practice any patented invention without a license or in any way infringe upon the intellectual property rights of any other party.

V. CONCLUSION

This project deals with complete hot runner mould tool design as per the parameters provided. Core and cavity is removed for placing the cap. Application used for designing this model, core-cavity and die design is Pro-Engineering application. Mould Flow Analysis is done on cap set which is made for the cavity preparation. We are using mould flow analysis for finding the material filling, pressure distribution, air traps, and weld lines formed during injection molding process. Mould Flow Analysis is done using "Plastic Advisor" which is a module in Pro/Engineer.

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. Hot runner mould is used for minimizing expenditure and cost estimation. 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 process manufacture of caps can be done without any failure.

VI. FUTURE SCOPE

In future hot runner technology may implement in large casting molds for the smaller components to do the massive production. In plastic molds we can make more quality and massive production within low cost Composite materials can be implemented in the manufacturing of molds to reduce cost and to increase life cycle time.

REFERENCE

[1] Plastics Technology, October 2005, "No. 1- Reciprocating-Screw Injection" US Patent 2,705,342 Hendry Apr 5, 1955

[2] F. Shi, Optimisation of Plastic Injection Moulding Process with Soft Computing, The International Journal of Advanced Manufacturing Technology, (2003) 21:656-661 Springer Verlag London Limited

[3] E. Bociga and T. Jaruga, Experimental investigation of polymer flow in injection mould.

[4] David O.Kazmer, Assistant Professor, University of Massachusetts, Polymer Injection Molding Technology for the Next Millennium.

[5] D. Mathivanan, M. Nouby and R. Vidhya, Minimization of sink mark defects in injection molding process.

[6] Dr. Martin P. Jones, Dr. Richard N. Callahan, and Dr. Richard D. Bruce, Flow Length Measurement of Injection Molded Spirals Using a Flatbed Scanner.

[7] X. Brouwers, P.Miniou. European Technical Center , Hot runner manifold nozzle and gate design consideration for successful molding of semi-crystalline polymers

[8] Darshan.P& Ramesh Babu. K, Mold flow analysis for PH meter cover Reference

[9] Ashutosh Modi, Prof. K.A. Rade, S.S. Kulkarni, Design enhancement for an injection Mold inputs from flow analysis.