ISSN 2393-8471

International Journal of Recent Research in Civil and Mechanical Engineering (IJRRCME)
Vol. 3, Issue 1, pp: (17-23), Month: April 2016 – September 2016, Available at: www.paperpublications.org

Design & Development of Injection Mold Using Flow Analysis and Higher End Design Software

Mohammad Safi A. Pathan[1], Kulkarni Pooja J[2], Jadhav Ramdas S[3], Lalit N. Patil[4]

Department of Mechanical Engineering, Gokhale Education Society's, R.H.Sapat College of Engineering, Management Studies and Research, Nashik, Maharashtra, India

Abstract: **In past Mold Design process was time taking as well as hectic. At first Drawing board, then 2D software's were used after which patterns were made. But in this case the results were not considerable most of the times. Thus the design to market time increased immensely and also project cost required was on a higher side.**

Due to the technological advancement the process of Mold Design has fastened and also the results are considerable. With the help of 3D software we can create Parametric Design, Which are editable. Also we can look at number of possibilities for designing a mold. Most importantly the process of Drawing Creation for Mold Design becomes very easy. In a 3D software Visualization of our design is easy possible. Thus our project aims at the awareness of developments of the new age technology of 3D CAD/Mold Wizard for Mold Design.

Keywords: **Progressive Die Design.**

1. INTRODUCTION

The molding may cause defects and its processing offers a challenge during its advance phase. The mold cost is high and any process that is not Improved renders heavy overheads during its development cycle and production. So designing the mold which confirms best suitability for the features on the component with even flow of molten plastic is very important part of development process.

The effective launch of any plastic product be subject to on knowing the true costs and profitability before the job is started. Injection molding normally involves large volumes of parts. Small cost outflows per part can be compounded to large cost differences over the life span of the part. Major cost components considered are material, re-grind and machine costs. Scrap, rejections and regrind costs are also considered in the cost.

Generally, plastic injection molding design contains a plastic part design, a mold design, and a design of injection molding process, all of which subsidize to the quality of the molded product and the production efficiency. The developed program system makes possible to perform: 3D modeling of the parts, numerical simulation of injection molding, analysis of part designs, and mold design with calculation. By the realization of the projected integrated system, this problem could be solved. The main part of the system consists of a knowledge based.

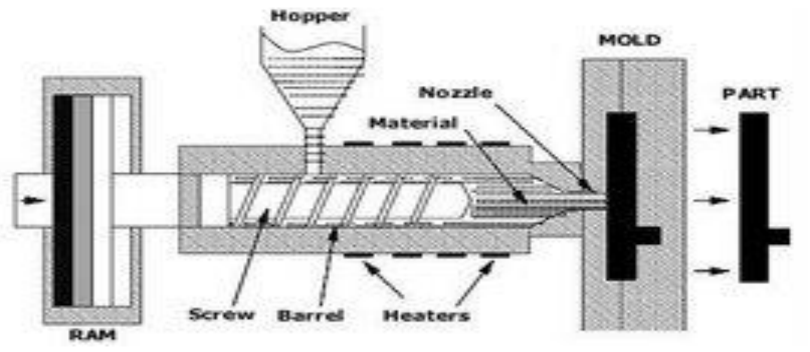


Fig. 1. Injection Molding Process

2. CASE STUDY

In the past molds there were only four cavities for spoon, which consumes too much time for large production as per requirements. This also leads to fatigue to the operators in the company & overall rate of production goes on increasing leading to loss to the manufacturer. Hence we have decided to design the mold which carries 12 cavities for spoon, due to which the production rate per hour goes double than previous production rate, which can also leads to overall profit of the company.

In The old mold (4 cavity mold):

Mold closing Opening time (2 M) = 3sec

Injection time (T) =3 sec

Cooling time (C) =10 sec

Ejection (E) = 4 sec

Cycle time Formula=2M+T+C+E=20

Cycle time=20 sec

Pressure control= depends on pressure gauge

Production per day (Max 10 Hrs/Day) =

1800 X 4 shots = 7200 no's

Weight of Mold= 110 kg

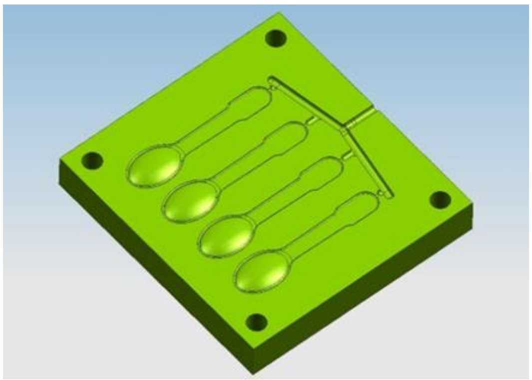


Fig.2. Mold for 4 cavities

In The New mold (12 cavity Mold):

Mold closing/Opening time (2 M) = 4sec

Injection Time (T) =5 sec

Cooling time (C) =14 sec

Ejection (E) = 3 sec

Cycle time Formula=2M+T+C+E=26

Cycle time=26 sec

Pressure control= depends on pressure gauge

Production per day **(**Max 10 Hrs/Day**) =**

1380 X 12 shots = 16,560 no's

Weight of Mold= 350 kg

International Journal of Recent Research in Civil and Mechanical Engineering (IJRRCME)
Vol. 3, Issue 1, pp: (17-23), Month: April 2016 – September 2016, Available at: www.paperpublications.org

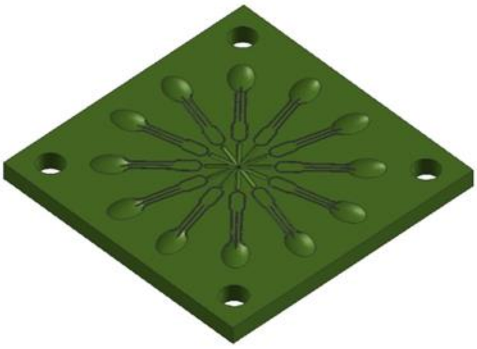
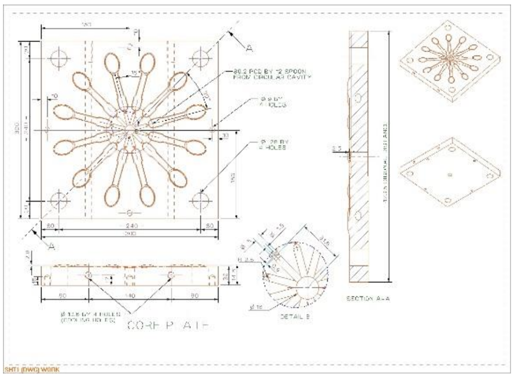


Fig.3. Mold for 12 cavities.



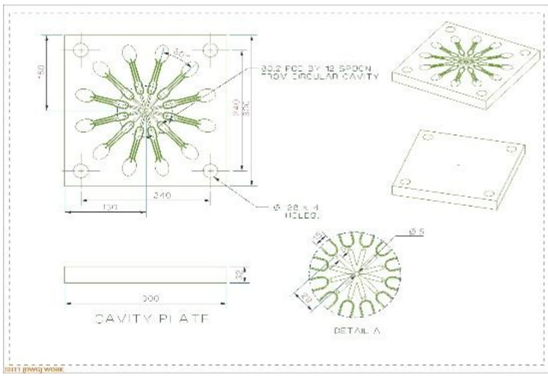


Fig.4. Schematic for Core plate **Fig.5. Schematic for Cavity plate**

Material Selection for Mold-

Core and Cavity –OHNS (Oil Hardening Non Shrinking).

Density - 8670 kg/m3

Hardness - (60-64) kg/mm2

Stock Remover Plate

Function:- To remove Stock from the core (female plate) after the casting operation is done

Material:- MS

C-35 = Steel for low stressed part

Properties -: C35 (mild steel) St32.

Tensile strength = 620 – 440 N/mm^2

Yield strength = 490 – 780 N/mm^2

Specific Weight = 0.0725 N/cc

Dimensions are:-

Length = 300 mm

Width = 300 -90 = 210 mm

Thickness = 35 mm

Remover pin are arranged in circular manner, as there are twelve spoon cavities and each of them are arranged at 30^0 angles. The ejector pin is considered as of 3mm diameter. Total no. of pin is 12 arranged circular manners which are attached to stock remover plate and at the time of process it is inserted in cavity plate to remove finished product.

International Journal of Recent Research in Civil and Mechanical Engineering (IJRRCME)
Vol. 3, Issue 1, pp: (17-23), Month: April 2016 – September 2016, Available at: www.paperpublications.org

Component Name:- Support plate

Support plates are plate such as stock remover plate, top plate, bottom plate.

Material:- Mild Steel (C35).

Tensile Strength 620 N/mm^2

Yield Strength 500 N/mm^2

Compression Strength 930 N/mm^2

Shear Strength 310 N/mm^2

3. MATERIAL FOR SPOON

Polystyrene (PS): Processes high Gloss sparker and transparence. Has low water absorption and moderate electrical insulation. It can be colored by adding dry colours. The moldings can be decorated by screen printing, painting and hot foiling. Shrinkage: 0.005-0.007

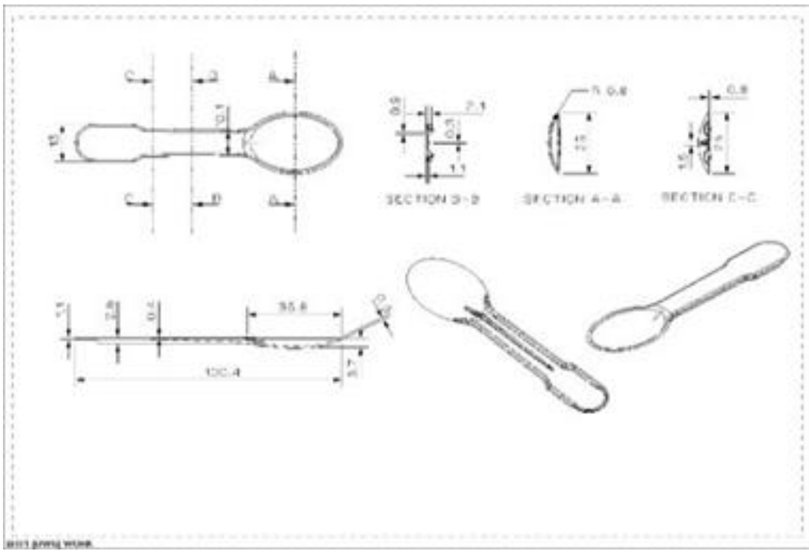


Fig.6. Schematic design Of Spoon

Resin	Shrinkage mm/mm
ABS	0.004-0.005
Acetal	0.015-0.025
Acrylic	0.003-0.006
HDPE	0.020-0.030
LDPE	0.015-0.030
Nylon	0.010-0.020
PC	0.006-0.008
PP	0.012-0.020
PS	0.005-0.007
SAN	0.004-0.006

Fig.7. Shrinkage Chart

All the dimensions are taken practically with the help of digital micrometer.

By considering the shrinkage property of spoon material.

Dimension of single Spoon Cavity

Top Width = 25.5 mm

Bottom Width = 15.5 mm

Length = 101 mm

In arrangement of the Twelve Spoon,

Each Spoon is Separated from each other is at 30^0 angle measured from the axis of spoons.

Component Name: - Riser

Riser is located at the bottom of the spoons

Thickness of each riser is Consider as 5mm.

Component Name: - Cavity Plate

Thickness of Cavity Plate.

New Cavity Plate Which is for twelve spoon is considered as 32mm.

Tc = 32mm

Thickness of required product Tp of Spoon is calculated as 3.5 mm.

Tp= 3.5mm.

Clearance Between the spoon cavity of end edge of cavity plate.

From Dimension of Twelve spoon arrangement the total length is covered is 264 mm.

Therefore Clearance is considered as 18 mm

The total length of Cavity Plate = Total length covered by spoon cavity + 2 (clearance length).

Lc = 300 mm

The dimensions of cavity plate are,

Length = 300 mm

Thickness = 32 mm

Width = 300mm

Component Name:- Top Plate

Material:- MS

Thickness of top plate

Tt = 1.15 Tc

Tt =1.15 x 32

Tt = 36mm

The dimension of top plate are,

International Journal of Recent Research in Civil and Mechanical Engineering (IJRRCME)
Vol. 3, Issue 1, pp: (17-23), Month: April 2016 – September 2016, Available at: www.paperpublications.org

Length = 300mm

Thickness = 36mm

Width = 300mm

Clamping Force:

The clamping force required to keep the mold closed during injection must exceed the force given by the product of the opening pressure in the cavity and the total projected area of all impressions and runners. Lower clamping values can be used with screw presses owing to the lower injection pressures possible with these machines.

$$\text{Clamping force (Tons)} = \begin{bmatrix} \Pr oject\, area\, of \\ moulding/s\,(Sq.Cm) \\ including\, runners \end{bmatrix} x \begin{bmatrix} \frac{1}{2} to \frac{1}{3} of\ Injection\, pressure \\ (Tons/sq.Cm.) \end{bmatrix}$$

65 tons= [PM] x {1/2 of 33.39 Mpa}

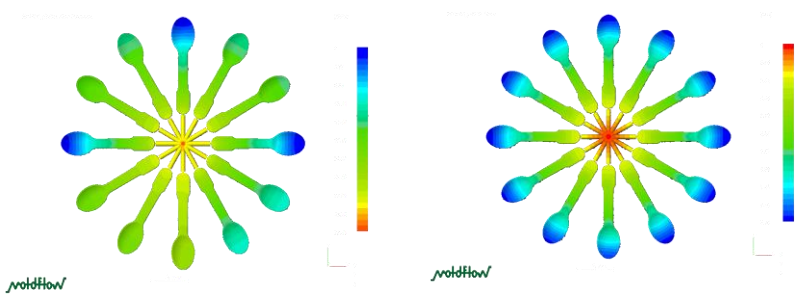
Project area of molding = 214 mm^2.

4. FLOW ANALYSIS

The “flow analysis” of the component would provide useful inputs for expecting the performance of component during its processing phase. It is generally not achievable to generate a soft mold for testing because of high cost involved.

Variations over the mold design will be done by changing the parameters like type of gate, gating system position, venting location and location of runners and risers for producing the fault free component. These parameters will be changed at least in three levels and suitable experimentation method will be followed. From the simulation and analysis, Mold flow software provides enough information regarding its filling time, injection pressure and pressure drop.

With these results, users can avoid the defect of the plastic in actual injection such as sink spot, hesitation, and air traps. The analysis will also help the mold designer to design a perfect mold with least modifications and which will also reduce the mold arrangement time. With this analysis and simulation, it will help to reduce time and cost. It is used generally by the plastics injection molding industry. The MOLDFLOW injection molding Simulation of polymers can provide information on the thermo-mechanical properties and residual stresses of the Part causing from the manufacturing process. MOLDFLOW writes this information to a boundary file for subsequent finite element stress analysis.



FILL TIME= 0.54 SEC **INJECTION PRESSURE=33.69 MPA**

Fig.8. Flow Analysis

5. CONCLUSION

The Design of the Mold and the processing factors has an effect over the quality of the part produced. Defects can be reduced through improved design of the mold with the study of simulation of flow through the mold. The material, size, obscurity (complexity) and the rate of production required should be considered for evolving the right Mold design for the given component. From the analysis simulation, Mold flow provides acceptable information results such as fill time, injection pressure and pressure drop. With this result, users can avoid the imperfection of the plastic in actual injection such as sink mark, indecision, over packing, air traps. The analysis will also help the mold designer to design a perfect

ISSN 2393-8471

International Journal of Recent Research in Civil and Mechanical Engineering (IJRRCME)
Vol. 3, Issue 1, pp: (17-23), Month: April 2016 – September 2016, Available at: www.paperpublications.org

mold with minimum modifications and it will also reduce the mold setup time. With this analysis and simulation it will help to lessen time and cost.

REFERENCES

[1] 'Plastic Injection Molding Process' Pertanika J. Science. & Technology. Supplement (2004) [85-99].

[2] Wang Li-tao, Zhou Chun-ying, 'Injection Mold Design built on Plastic Advisor Analysis Software in Pro/E,' IEEE 2011.

[3] P.K. Bharti, (2010) "Recent Methods for Optimization of Plastic Injection Molding Process, A Retrospective and Literature Review', IJEST.

[4] Min B.H., (2003), "A study on quality monitoring of injection-molded parts". Resources Processing Technology, pp. 1-6.

[5] ROSATO, Injection Molding Handbook. Applied Publisher Ltd. Essex. (1986).

[6] S. H. Wang. Teaching System of "Mold design and manufacturing" in higher vocational colleges major constructed on regional economic development. Journal of Liuzhou Vocational and Technical College 2009; 9:41-5.

[7] 'Misumi' Standard Components For Plastic Mold-Misumi India Pvt. Ltd.(2012)