Design and Manufacture of Injection Mold for Flat Drippers using Hot Drop Technology

Atharva Suyog Mohole
Department of Mechanical Engineering
K.K. Wagh Institute of Engineering Education and Research, India

Abstract— Drip irrigation impacts prominently in proposed solutions to the water crisis. Earlier Inline and Outline drippers were used which had comparatively higher material cost & lower efficiency. Later, Round Drippers came into use which had Adequate efficiency but higher material cost because of the bigger size. So as a solution to this lately Flat Drippers have been introduced. Flat Drippers have very small size hence less material cost. Flat Drippers are manufactured using Plastic Injection Molding Process. Plastic Injection Molding is a process of melting plastic pellets (thermosetting/thermoplastic polymers) that once malleable enough, are injected at pressure into a mold cavity, which fills and solidifies to produce a product. Flat Drippers with zero runner wastage, zero rejections and the least possible cycle time. Hence, increasing the Production rate as well as it maintains the quality and consistency of the product. In the scope of this project the main aim is to Design and Manufacture Injection Mold (Hot Runner) for Flat Drippers to produce a standard discharge of 4 liters of water per hour in a single dripper. So, we will design and manufacture a mold of 32 cavity with 2 hot drops in such a way that the dripper meets the standard discharge parameters.

Keywords: Flat Dripper, Injection Molding, Hot Drop, Cycle Time

I. INTRODUCTION

Injection molding is an ancient technology that has been used since the late 1800's. Injection molding machines incorporate a huge screw to force molten plastic into the mold at high pressure. This screw drive method was invented in 1946 and is still the method used today. Injection molding machines definitely do not have the modern, high-tech feel of 3D printing technology. There is really nothing cool about injection molding, but nonetheless it is a requirement for most hardware products. An injection mold consists of two halves that are forced together to form a cavity in the shape of the part to be produced. Hot, liquid plastic is then injected at high pressure into this cavity. The high pressure is needed to ensure that the plastic resin fills in every crook and cranny of the mold cavity. Once the plastic has had time to cool, the two halves of the mold are pulled apart, and the part is ejected.

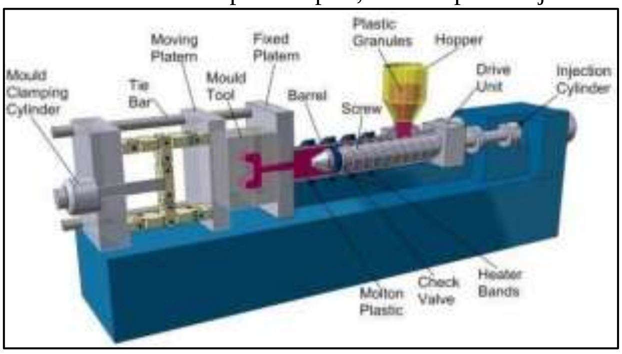

Fig. 1: Injection Molding Machine

Although designing for injection molding can be quite complicated, and the cost of the molds themselves are incredibly expensive, there is one huge reason why injection molding is still used today. No technology can beat injection molding when it comes to producing millions of identical copies of a part at an incredibly low price. Injection molds are expensive, and you'll most likely need a few of them, so their total cost can be quite significant. The more parts you need to produce with the mold the more expensive the mold. This is because the mold must be designed to withstand incredibly harsh conditions. Over and over again a mold is subjected to high temperature and high pressure.

II. HISTORY OF MOLD

Mold making has existed for thousands of years. Molds have been found around the world from ancient times including a stone mold dating back to the bronze age used to make spear tips to a mold from ancient Greece that was used to make figurines and even a wooden mold to make candy in Sri Lanka. The history of mold making shows molds were used across cultures for defense, hunting, and leisure, so ancient civilizations understood their value early on. Injection molding traces its roots back to more recent times. The first recorded event for injection molding is 1872 when John Wesley Hyatt and his brother Isaiah patented the first injection molding machine. According to Wikipedia, in 1903 German chemists Arthur Eichengrün and Theodore Becker invented the first soluble forms of cellulose acetate, which can be used to make items such as eyeglass frames, and Eichengrün developed the first injection molding press in 1919.

As a natural follow-on, Eichengrün patented the injection molding of plasticized cellulose acetate in 1939. World War II created a huge demand for inexpensive, mass-produced products in the 1940s, so injection molding saw an increase in use. After World War II, in 1946, James Watson Hendry continued the momentum of injection molding when he built the first screw injection machine. Screw injection machines today are used in the majority of injection molding, showing that even though there have been few major innovations in the injection mold industry each innovation has meant a great deal to the industry. This machine allows more control over mold injection speed and the quality of items made using injection molding. It also allows for color to be added to the material being molded and mixed thoroughly before molding.

Hendry would then go on to develop the first gas-assisted injection molding process in the 1970s, which allowed the production of complex, hollow articles that cooled quickly. This led to greater design flexibility as well as better strength and finish of manufactured parts. Because of the rapid cooling the process also reduced production time, cost, weight and waste. By 1979, plastic production overtook steel production. The Tool room began producing superior quality plastic injection molds in 1978 and has industry experience dating all the way back to 1946, so has seen the plastic injection molding industry through the past half-century-plus of innovation and change. With its state-of-the-art molding facility and injection molding processes. The Tool room is the St. Louis area's leading plastic injection molding shop.

III. WORKING OF INJECTION MOLD

Working of injection molding is similar to extrusion and it works like an injection as name suggests. Molding material/ raw material poured into the hopper by feeding device. After that molding material goes down under the action of gravity into the cylinder (barrel) as shown in the diagram. A circumferential heater which is located on the barrel is used to melt the material. When powder form of molding material goes into the barrel from the hopper it starts melting and a hydraulic ram or rotating screw pushes the material forward into the mold by applying some pressure. Molten plastic material is injected into a closed mold attached on the other side of the barrel; in this split mold is used. Molding material goes forward continuously by the rotating screw. Pressure applied by the hydraulic system. Injection pressure is generally 100-150 MPa. After injection, pressure is applied for some time or locked at the same position with some force.

After the whole process is done the parts manufactured are cooled sufficiently. Then mold is open, and some ejectors are used for proper removal of the part without damage. After removing the part, the mold is closed again. This process is very fast and automatically repeated. Here complex shape parts can be easily manufactured. The main parts of the Injection molding machine are material hopper, barrel, an injection ram/ rotating screw type plunger, heating device (heater), movable pattern, ejectors, and a mold inside mold cavity.

Generally, injection molding machines work in a horizontal manner. Injection molding machine consists of a barrel (cylindrical pipe). A hopper is located at one end of the barrel. A hydraulic ram or rotating screw runs by an electric motor used to provide force is located inside the barrel. A heating element (heater) is attached to the barrel, circumstantially used to melt the molding material coming down from the hopper.

On the other side of the barrel a mold cavity is attached. Mold is located inside the mold cavity and a movable pattern is used in whole manufacturing. Mold is generally made up of copper, aluminum and tool steels. The life cycle for different material molds is different. This can be selected as per requirement.

IV. ADVANTAGES

1) The main advantage of this process is that complex shapes components having small wall thickness (5-15 mm) can be easily molded and removed from the die without damage.
2) Parts which are made by injection molding have good dimensional tolerance.
3) The major advantage of this technique is that the scrap produced by this is very less as compared to some other processes.
4) Parts made by injection molding process are competing with parts made by investment casting and complex machining parts.
5) This process has a high production rate as compared to other techniques.

V. PROBLEM STATEMENT

Rejection of flat drippers due to short molding caused due to injection through a single injection point and long runner path also resulting in increased cycle time and runner wastage ultimately affecting production rate and increasing customer complaints. When the product was manufactured on the old mold, we observed the process carefully and noticed some major drawbacks.
1) Short molding
2) Flashes
3) Material wastage
4) Longer runner path
5) Increased cycle time
6) Low production rate
7) Profit margins

VI. OBJECTIVES

1) To increase production rate of flat drippers
2) To Reduce material wastage to zero by providing individual injection points for each dripper cavity.
3) To Reduce cycle time by giving individual injections for each component.

VII. SCOPE

1) To use hot runner (hot drop technology) to increase production rate
2) To study the results obtained.
3) To maximize the production rate and to shorten the cycle time.
4) Less defective products.

VIII. METHODOLOGY

1) Literature survey on injection molding.
2) Analyzing the old mold.
3) Identifying all the problems of the previous mold.
4) Design development of new mold.
5) Analysis of performance of the new mold with respect to production rate, cycle time, material wastage.
6) Testing of new mold.

IX. LITERATURE REVIEW

Injection molding is the most traditional plastic parts manufacturing procedure. Using injection molding, a large range of items are made, varying in scale, complexity and implementation. Hot runner is better than cold runner but is seldom utilized due to high cost and difficulty. Every substance needs a complicated set of parameters such as injection temperature, injection pressure, flow rate, mold temperature, ejection temperature, cooling rate and cycle time. Improper set of parameters leads to many flow lines, burn marks, warping, vacuum voids / air pockets, sink openings, weld lines, jetting. Few defects, including discoloration, plastic use and delamination storage. Quick shots and flash triggers faulty construction or fix. These criteria include process optimization and defect-removal collection. The impact of these criteria on the molding process and recommendations to generate defect- free components needs more study.

Mr. P. Vinod, Mr. K. Vijay Kumar have designed multi-cavity injection mold with HRS and CRS. By comparing both designs, they researched the impact of runner systems, mold cooling and venting. Molding analysis is carried out using ANSYS. G. Rajendra Prasad, Dr S. Chakradhar Goud Studied dynamic characteristics like hot runner nozzle strain using analysis in FEA. N. Diva, Dr. S. N. Malleswara Rao, Dr. V. S. Parameswara Rao, concluded in their paper that the hot runner system accommodates the molten plastic. The runner method determines mold component consistency and productivity. They performed structural and thermal analysis of the mold's original and updated designs concluding that the modified design produces the best performance. Rishi. A. Yadav, S. V. Joshi, N. K. Kamble, recent studies to design and determine injection molding method parameters. In the parameter environment for injection molding, several test works were performed focused on various approaches. In the plastic injection molding (PIM) industry, optimum process parameter settings are critically affected by performance, consistency, and cost of output. A. Demirer, Y. Soydan, unlike the conventional runner approach, hot runner machine effects on injection molding method and injection product properties were studied. They used data from method parameters experiments. Injection pressure and temperature change a broad variety. For the hot runner process, injection pressure was marginally lower for higher weight samples. If the temperature of the process increases, shrinkage and warpage increases, reducing with increased injection pressure and happening at a low level where the sample weight was high. Gurjeet Singh, Ajay Verma Studied primary molding conditions from design creation to product manufacturing. They studied different factors based on processing parameters. It is concluded that efficiency declines when channelizing efforts to improve quality. Parameters must be optimized to ensure good quality and efficiency. Authors analyzed different responses to injection molding process quality based on output parameters and methods. Mehdi Moayyedian, Kazem Abhary, the injection molding method implemented a new gate geometry. It was observed that current edge gate corners create turbulence of molten plastic leading to internal and external defects.

New geometry was introduced to reduce injected parts' internal and external flaws. The study's goal was to make the final piece easier geometry, which eliminates the last part's apparent blemish after de-gating. V.Chandra Sekhar, N. Jaya Krishana Suggested design for two circular flat plate 1 mm wide. Contribution of this research was to change the existing edge gate geometry by eliminating rectangular edge gate corners to minimize scrap occurrence of injected bits. Smooth plastic flow through cavities often prevents internal and external defects. The result reveals no shot-filling cavities. No weld lines, meld lines or sink marks were identified with new edge gate design. The experiment ends with an added portion of the initial edge gate step, less noticeable than the current edge gate.

Harold Godwin, Mold Making Technology — Competition and price pressure in the injection molding industry affected many manufacturers of OEMs, molders, and molds. The required operating costs were due to increased resin, electricity, human capital, resulting in profit pressure. The decision to use a hot runner was noticeable for medium and large molded component volumes. Sal Benenati, Hot runner systems are very popular injection molds. Hot runner suppliers must adjust part designs to satisfy more strict output and material demands, sometimes adding complexity to products. Some techniques, for example, use moving motors to control valve gates, some need specific heaters, and some require unique mold tolerances to function properly. Standard convection, low shear velocity, and high viscosity-dependent effects were studied. Lee and Lin et al. built a multi-cavity mold runner and gating system. Use the Finite Element Theory (FEM) network. Optimal runner unit parameter used to minimize injection mold warp, FEM, Taguchi phase and adductive framework. Processes during mold filling, enhancing moulding condition. Model injection mold simulation at steady flow rate. Finite differences method offers strong consent for methodological solutions. Different gate sizes and locations using flow simulation have been detected for defects reduction such as weld lines and air traps, air traps and war page can be managed by varying process parameters. Part flow-reducer studied using Autodesk Mold Flow tools. The mold flow analysis is used to predict the piece's deformation and change the design accordingly. The plastic toy building block section is analyzed; cycle time is successful for four cavities as filling and cooling time for four cavities does not improve The outcome indicates that both parameters influence product consistency, Plastic Flow Simulation analyses the flow of molten plastic to optimize part and mold designs, reducing possible part defects. Simulation research was undertaken for optimum gate location. Comparing two gate positions was achieved. Comparative analysis found the optimal gate location. Several product flaws decrease due to improper eating. Comparative analysis of the use of different gate types and runner systems for the same job, resulting in a satisfactory reduction of molding defects and increased pressure in the component, which is also within limits. Vikas B J and Chandra Kumar R Optimizing scrap processing period. Manual GUI plays a key role in producing plastic components without compromising product consistency. This paper explains gate position and size effects by repetitive analyses. Plastic flow advisor programme prevents fill, scrap and automated degassing. Simulation analyses method parameters including fill time, shrinkage, weld lines, pressure decrease, and air traps in successive tests. Experimental verification for current improved gate location in the injection molding method. Reduced shrinkage and ventilation, minimizing defeats. Simulation software has greatly impacted the injection molding industry. Cheaper simulation simulations are an alternative to traditional factory-floor experiments. Study on developing plastic injection molding method has now expanded a tonne. Sadeghi's Built a neural network model to predict consistency or soundness of injected plastic parts based on major process variables and differing material grade. The system shortened preparation period, improving process requirements or operational parameters. Lee and Kim used a revised, complicated war page reduction method by optimizing various surface thicknesses, further growing the war page by reaching optimal processing conditions. Ozcelik and Erzurumlu Comparison acrylic war page molding for ANOVA. Neural, genetic algorithm. Sapura. The results of the injection molding system and simulation analysis of MPI software were contrasted with those of a limited manufacturing process MPI was found to measure the optimum pressure required to correctly fill the mold cavity and graphically track weld lines. MPI offered further simulation analysis.

X. CALCULATIONS

Runner Diameter $(D) = ((W^{1/2}) * (L^{1/4})) / 3.7$
… (R.G.W. Pye Book)
where,
W = weight of component in gm
L = length of runner in mm
Here,
W = 0.000503 kg = 0.503 gm
L = 150 mm
So, $D = ((0.503*16)^{1/2} * (150)^{1/4}) / 3.7$ $D = ((8.048)^{1/2} * (3.499)) / 3.7$
$D = 2.683$ mm
Since D is less than 3 mm, we use modified formula,
$D = (((W)^{1/2} + (L)^{1/4})) / 3.7) + k$
$D = 2.683 + 0.06$ … (when $D<3$, we consider $k=0.06$) $D = 2.741$ mm
Therefore, $D \approx 2.75$ mm (for safety)

XI. EXISTING CONVENTIONAL MOLDS

The old mold had a single injection point which led to incomplete filling of cavities. It also had a longer runner path which resulted in increased cycle time and material wastage. In this mold, hot drop technology is not used.

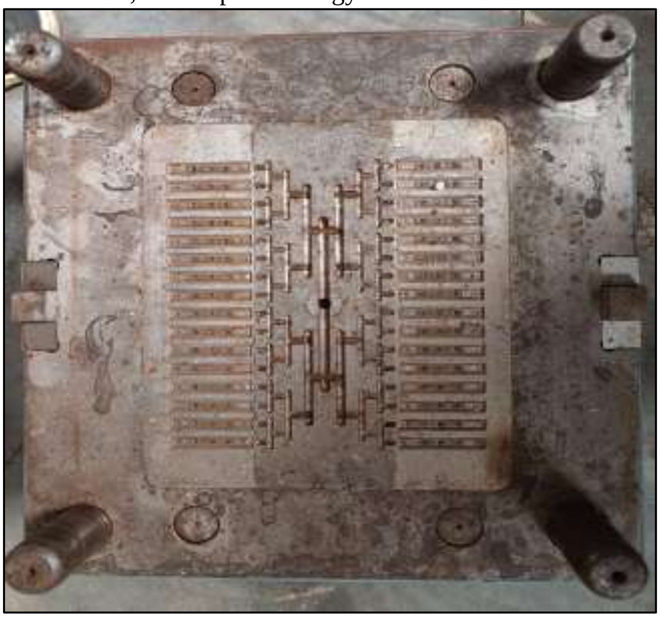

Fig. 2: Conventional Mold

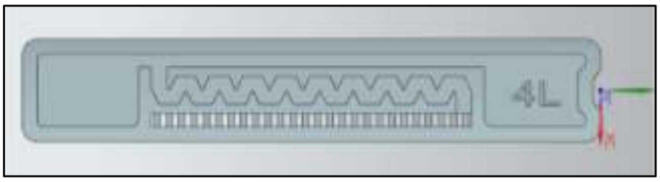

Fig. 3: Flat Dripper Top View

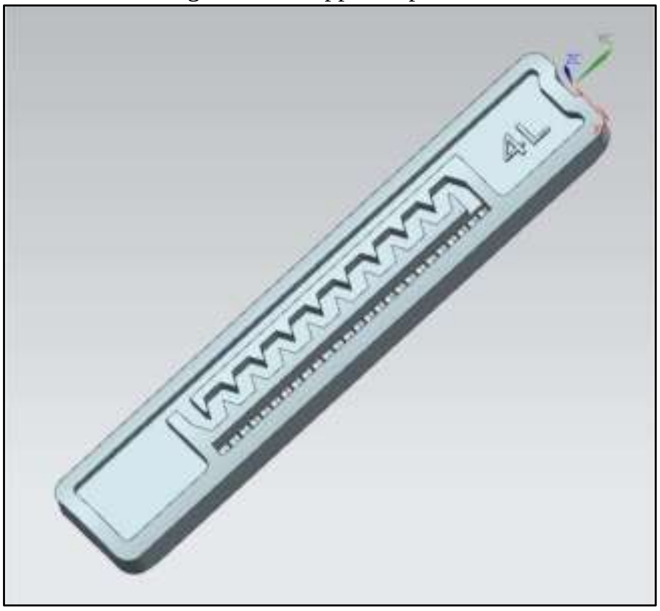

Fig. 4: Flat Dripper 3D View

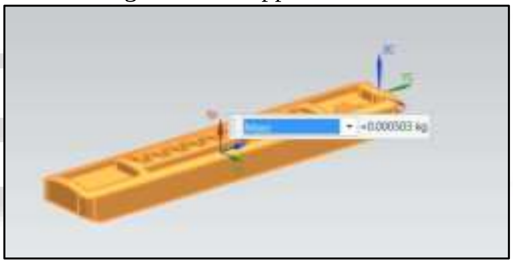

Fig. 5: Mass of Flat Dripper

XII. DESIGN AND DIMENSIONS OF CORE, CAVITY, CORE HOUSING, CAVITY HOUSING

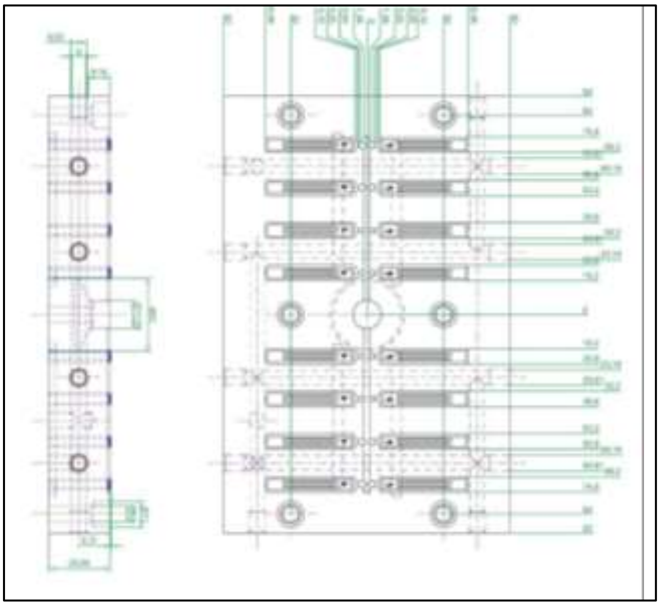

Fig. 6: Cavity Block 2D View

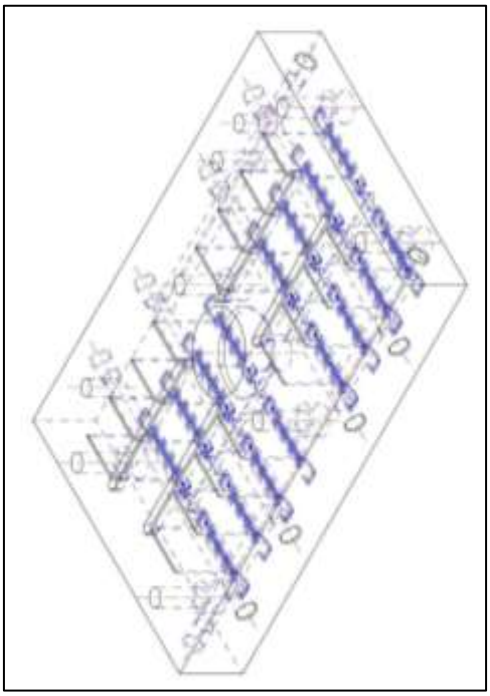

Fig. 7: Cavity Block Isometric View

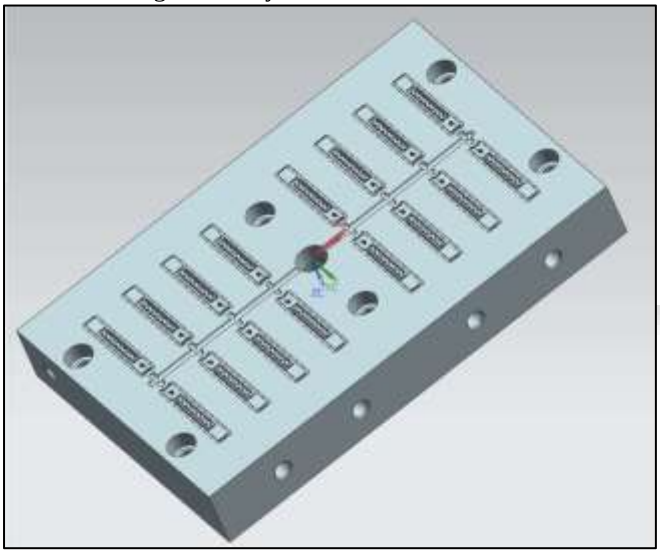

Fig. 8: Cavity Block 3D Top View

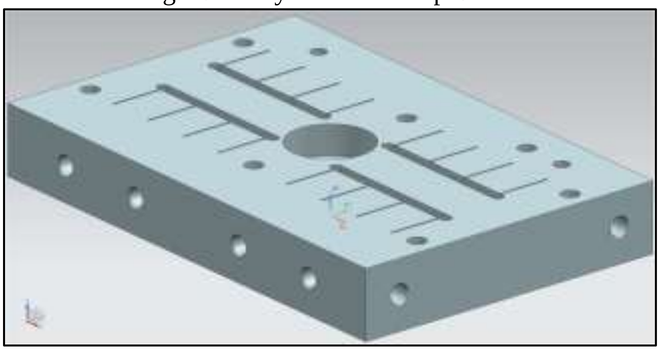

Fig. 9: Cavity Block 3D Bottom View

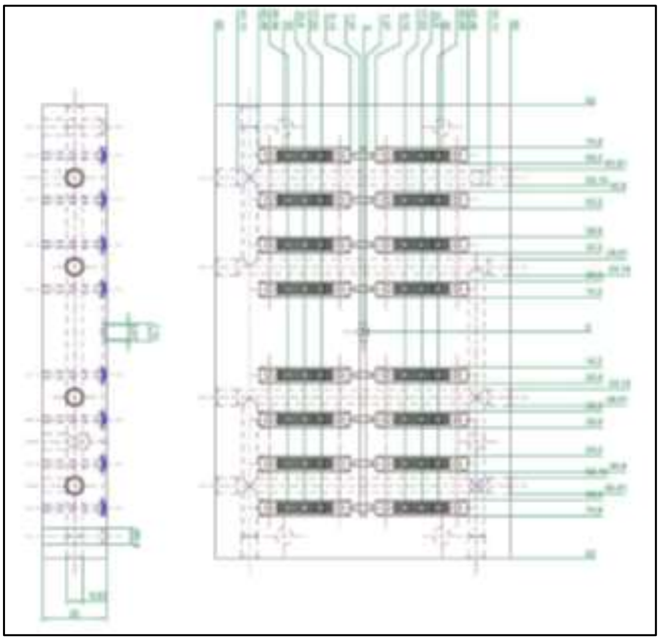

Fig. 10: Core Block 2D View

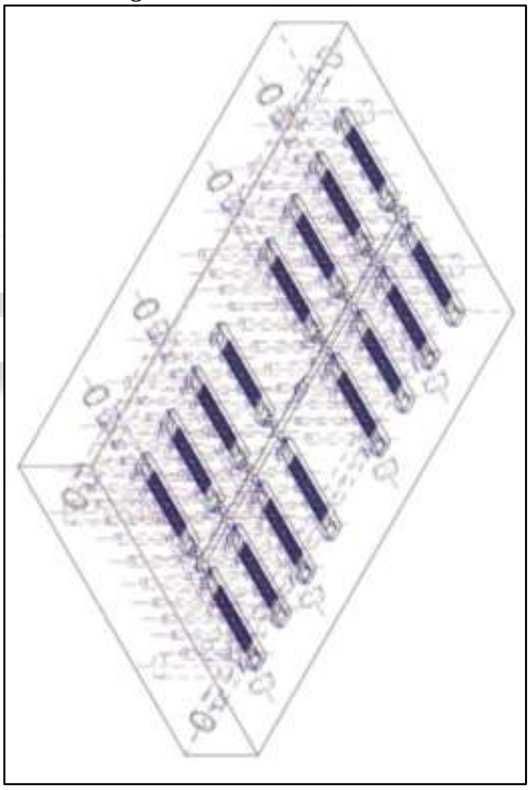

Fig. 11: Core Block Isometric View

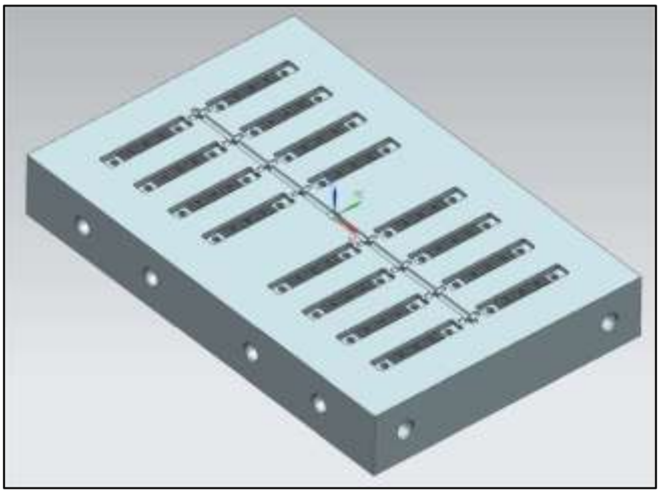

Fig. 12: Core Block 3D Top View

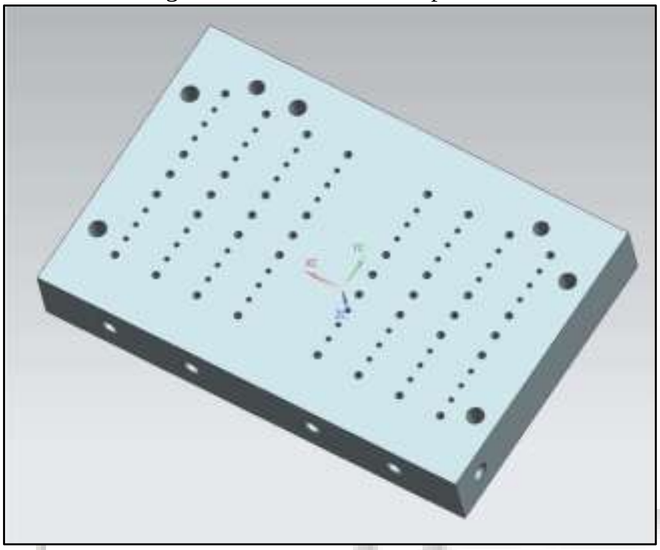

Fig. 13: Core Block 3D Bottom View

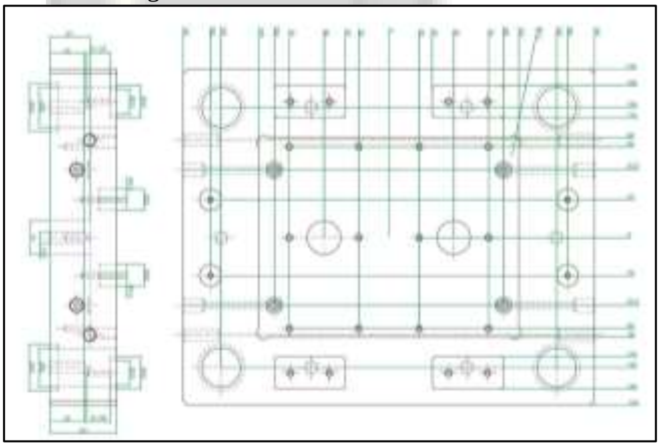

Fig. 14: Cavity Housing 2D View

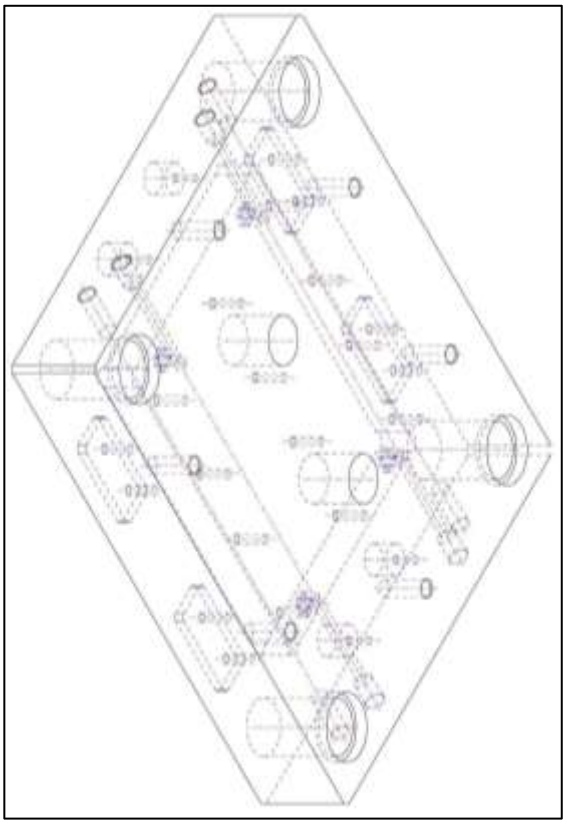

Fig. 15: Cavity Housing Isometric View

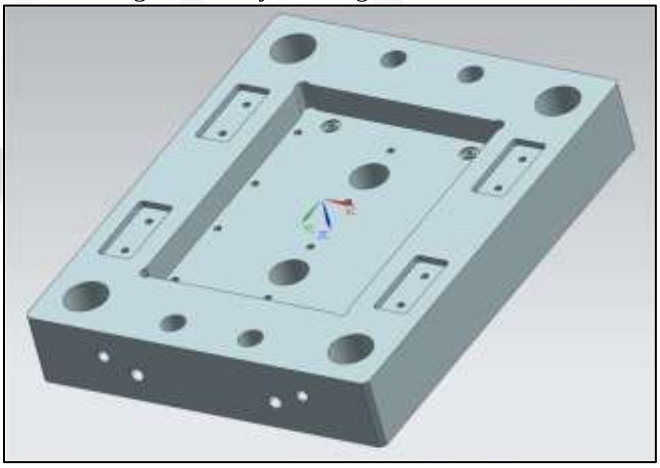

Fig. 16: Cavity Housing 3D Top View

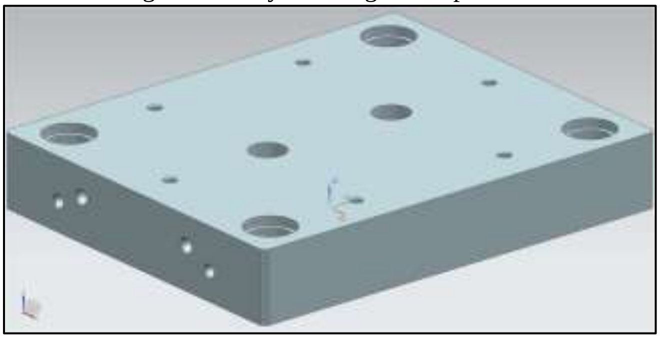

Fig. 17: Cavity Housing 3D Bottom View

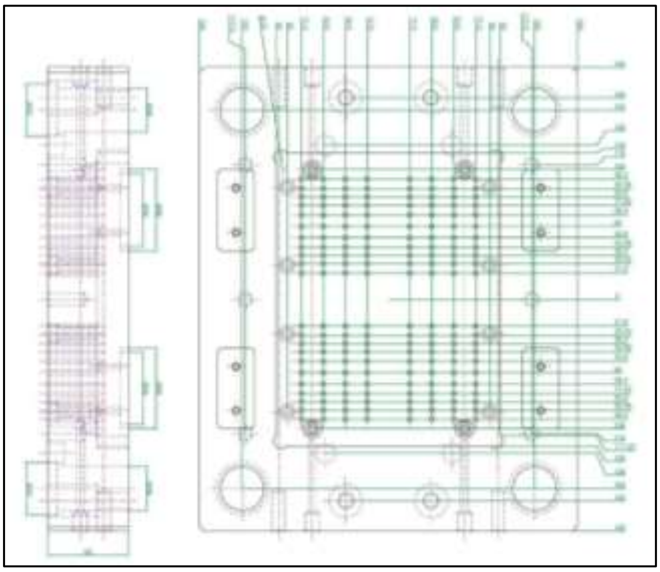

Fig. 18: Core Housing 2D View

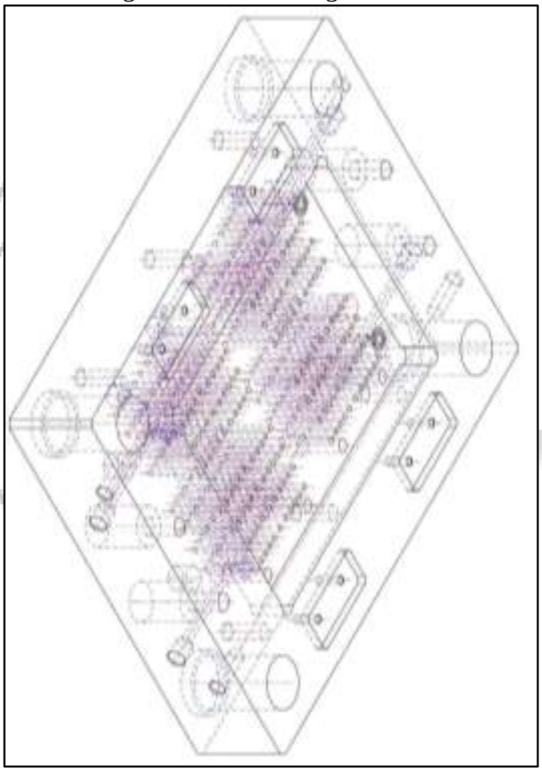

Fig. 19: Core Housing Isometric View

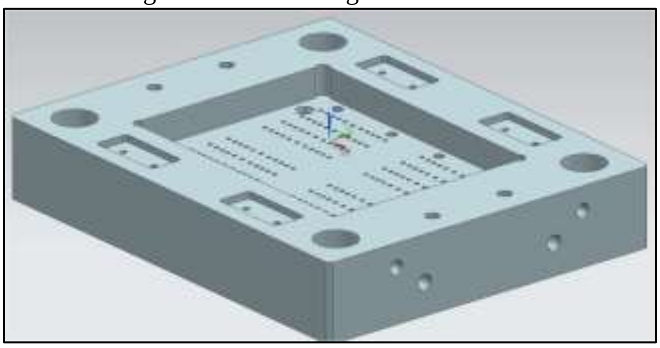

Fig. 20: Core Housing 3D Top View

Fig. 21: Core Housing 3D Bottom View

REFERENCES

[1] Harshal P. Kale, Umesh V. Hambire, "Optimization of injection moulding process for reducing shrinkage of high-density polyethylene (HDPE) material", International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, Volume 5, Issue-6, 2017.

[2] Junhui Liu, Xindu Chen, Zeqin Lin, and Shipu Diao, "Multi objective optimization of injection moulding process parameters for the precision manufacturing of plastic optical lens", Hindawi Mathematical Problems in Engineering, Article ID 2834013, 13 pages, 2017.

[3] D.Venkatesa Prabu, A. Sathishkumar, "Design Enhancement and Optimization of Plastic Injection Moulding", IJSRD- International Journal for Scientific Research & Development, ISSN: 2321-0613, Vol. 4, Issue 12, 2017.

[4] Sokkalingam Rajalingam, Pandian Vasant, "Optimization of Injection moulding process parameters for a plastic cell phone housing component", Published by the American Institute of Physics, AIP Conference Proceedings 1787, 080021, 2016.

[5] Jithin K, Kannakumar K, "Parameter Optimization of Injection Mould Using Concurrent Approach", IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 12, ISSN: 2348 - 7968, 2016.

[6] Mr. Aditya M. Darekar, Prof. T. S. Venkatesh, "Review of Optimization Aspects for Plastic Injection Moulding Process", RACST - Engineering Science and Technology: An International Journal (ESTIJ), ISSN: 2250-3498 Vol.5, 2015.

[7] http://unimouldasia.com/en/what-is-injection-moulding

