INJECTION MOULD TOOL DESIGN (TRIPPER PLATE) AND MOULD FLOW ANALYSIS OF EXHUST FAN

Mr.Kandula Yateesh Kumar[1] **Sri.K. Hemasundararao[2]**

[1]M.Tech (CAD/CAM) Student, Mechanical Engineering, Chirala Engineering College, Chirala, A.P, India
[2]Asistant Professor, Mechanical Engineering, Chirala Engineering College, Chirala, A.P, India

ABSTRACT

This paper presents the design of plastic injection mould for producing a plastic product. The plastic part was designed into two different types of product, but in the same usage function. One part is using clip function and another part is using tick function. In the computer-aided design (CAD), two plastic parts were drawn in 3 dimension (3D) view by using SIEMENS 8.0 parametric software. In the computer-aided manufacturing (CAM), DELCAM 12.0 software was used to develop the machining program. For mould design, the product was designed into two changeable inserts to produce two different types of plastic product in one mould base. Before proceeding to injection machine and mould design, this part was analyzed and simulated by using Ansys 15.0. From the analysis and simulation we can define the most suitable injection location, material temperature and pressure for injection. In the present work Component selected for conveyor chain locking links.

Keywords: Changeable insert mould, injection pressure, air traps, injection location, mould design .

1. INTRODUCTION

Because of the many inherent advantages in using plastic materials, there is an ongoing trend of replacing metal with injection-molded plastic parts in a wide variety of applications. More and more parts with critical end-use application requirements are becoming candidates for conversion to plastics. Plastics are lightweight, durable and corrosion-resistant; have a high strength-to-weight ratio; and, when used in transportation applications, for example, offer one of the easiest ways to increase fuel savings by making vehicles more lightweight. As plastics replace metals, the parts must be designed to take into account the properties of the specific plastic relative to the application requirements. One of the complicating factors for injection-molded plastic parts is that the properties of plastic materials effectively change during the manufacturing process. While this is not a problem in and of itself, problems can arise if the structural analyses are based on generic material data that does not accurately represent the actual properties of the molded part. These problems can include over engineering of components, which can lead to unnecessary costs and material usage, or under-engineering, which can result in part failure. Fiber-filled plastic materials are commonly used in metal replacement applications. When glass or carbon fibers are added to plastics, the elastic modulus can increase significantly with a negligible effect on part weight. This combination of low weight and high stiffness makes fiber-filled plastics ideal for high-performance applications. The key to unlocking the potential of these plastics lies in the orientation of the fibers. The orientation direction and the degree of orientation of the fibers determine the mechanical properties of the molded part. In areas where fibers are strongly aligned, the material will have higher strength characteristics in that direction, but will be relatively weak in the perpendicular direction (across the fibers). In areas where the fibers are more randomly oriented, the material will not achieve maximum strength, though the strength properties will not depend as much on the loading direction, creating a more isotropic like material.

INJECTION MOULDING:

Injection molding is the most common method of part manufacturing. It is a manufacturing process for producing parts for both thermoplastic and thermosetting plastic materials. The process is material is fed into a heated barrel from the hopper, mixed and forced into a mould cavity where it cools and hardens to the configuration of the mould cavity, the features of the desired part such as simple components to complex components. The advantages of injection moulding are high production rate, repeatability high tolerances, low labour cost and minimizes scrap cost but equipment investment and running cost is high.

EQUIPMENT

Injection moulding machine consist of a material hopper, screw type plunger or an injection ram and a heating unit. It holds the mould tool to get the required shape of the components. In which the amount of clamping force that the machine can exert in terms of in tonnage. The plastic material required more injection pressure to fill the mould and this more clamp tonnage required to hold the mould closed.

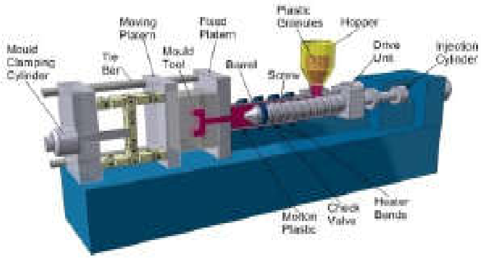

Fig. 1.1 Injection Moulding Machine

MOULD:

Mould or die are the common terms used to describe the tooling used to produce plastic parts in moulding. It is expensive to manufacturing and used in mass production. Generally the moulds are prepared by steel, aluminum and beryllium-copper alloy. The choice of material should be more parts have to be manufacturing before wearing out. Steel moulds are more cost to construct but their longer life span will offset the higher initial cost over a higher number of parts made before wearing. Moulds are manufacturing by the CNC milling machine. In this machine complex and more accurate moulds are prepared in less time; the selection of the milling tool bit depends on the operation and size of the work piece (mould).

TIME FUNCTION:

The time it takes to make a product using injection moulding can be calculated by adding:

wice the Mold Open/Close Time (2M) + Injection Time (T) + Cooling Time (C) + Ejection Time (E) Total time = 2M + T + C + E Total cycle time can be calculated by using t cycle = t mould close + t injection + t cooling + t mould open + t ejection. The time taken for mould (close, open) injection, ejecting are few seconds. The cooling times, which dominate the process which takes more time one.

LUBRICATION

The mould must be cooled in order for the reproduction to take place because of the heat capacity, inexpensiveness and more availability of water and is used as the primary cooling agent to cool the mould. It channeled through the mould to account for quick cooling times. A colder mould is more efficient because this allows for faster cycle times.

MOULD DESIGN

The mould consists of two primary components injection mould (plate A) and ejector mold (plate B). In which molten plastic enters into channels of mould through the injection barrel. Along these channels molten plastic enters into the various gates and into the cavity to form the desired shape of part. The amount of resins removed to fill the spruce, runner and cavity of a mould is a shot. Trapped air in the mould can escaped through air vents that are grinded into the parting line of the mould. Shrinkage is considered for determine the draft. The draft required for easy with drawl of the mould is dependent on the depth of cavity. If the skin is too thin, the mould cavities are touch to each other, the molten elastic will stick to the mould cavity. To avoid this and also to remove the molten part from the mould cavity ejector pin are used

2. METHODOLOGY

Uni-graphics software is one of the world's most advanced and tightly integrated

CAD/CAM/CAE software package developed by Siemens PLM Software, offers several pre-packaged Mach Series solutions for NC machining. Available in a range of capability levels, these solutions accelerate programming and improve productivity for a variety of typical manufacturing challenges, from basic machining to complex, multiple-axis and multi-function machining, as well as mould and die manufacturing it also merges solid and surface modeling techniques into one powerful tool set. The packages include complete capabilities for geometry import, CAD modeling and drafting, full associatively to part designs, NC tool path creation, verification and post processing, along with productivity tools that streamline the overall machining process.

CAM(COMPUTERAIDED MANUFACTURING)

Computer-aided manufacturing (CAM) is the use of computer software to control machine tools and related machinery in the manufacturing of work pieces. This is not the only definition for CAM, but it is the most common; CAM may also refer to the use of a computer to assist in all operations of a manufacturing plant, including planning, management, transportation and storage.[6][7] Its primary purpose is to create a faster production process and components and tooling with more precise dimensions and material consistency, which in some cases, uses only the required amount of raw material (thus minimizing waste), while simultaneously reducing energy consumption. CAM is now a system used in schools and lower educational purposes. CAM is a subsequent computer-aided process after computer-aided design (CAD) and sometimes computer-aided engineering (CAE), as the model generated in CAD and verified in CAE can be input into CAM software, which then controls the machine tool.

Traditionally, CAM has been considered as a numerical control (NC) programming tool, wherein two-dimensional (2-D) or three-dimensional (3-D) models of components generated in CAD software are used to generate G-code or M-code etc, which may be company/controller specific, to drive computer numerically controlled (CNC) machines. In modern day Controllers, CNC Machines, simple designs such as bolt circles or basic contours do not necessitate importing a CAD file for manufacturing operation.

Computer-integrated manufacturing (CIM) is the manufacturing approach of using computers to control the entire production process. This integration allows individual processes to exchange information with each other and initiate actions. Through the integration of computers, manufacturing can be faster and less error-prone, although the main advantage is the ability to create automated manufacturing processes. Typically CIM relies on closed-loop control processes, based on real-time input from sensors. It is also known as flexible design and manufacturing.

The term "computer-integrated manufacturing" is both a method of manufacturing and the name of a computer-automated system in which individual engineering, production, marketing, and support functions of a manufacturing enterprise are organized. In a CIM system functional areas such as design, analysis, planning, purchasing, cost accounting, inventory control, and distribution are linked through the computer with factory floor functions such as materials handling and management, providing direct control and monitoring of all the operations.

CNC MACHINING:

Numerical control (NC) is the automation of machine tools that are operated by precisely programmed commands encoded on a storage medium, as opposed to controlled manually via hand wheels or levers, or mechanically automated via cams alone. Most NC today is computer numerical control (CNC), in which computers play an integral part of the control.

In modern CNC systems, end-to-end component design is highly automated using computer-aided design (CAD) and computer-aided manufacturing (CAM) programs. The programs produce a computer file that is interpreted to extract the commands needed to operate a particular machine via a post processor, and then loaded into the CNC machines for production. Since any particular component might require the use of a number of different tools drills, saws, etc. Modern machines often combine multiple tools into a single "cell". In other installations, a number of different machines are used with an external controller and human or robotic operators that move the component from machine to machine. In either case, the series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.

CNC like systems are now used for any process that can be described as a series of movements and operations. These include laser cutting, welding, friction stir welding, ultrasonic welding, flame and plasma cutting, bending, spinning, hole-punching, pinning, gluing, fabric cutting, sewing, tape and fiber placement, routing, picking and placing (PnP), and sawing.

INJECTION MOULDING MACHINE

An injection moulding machine produces components by injection moulding process. Most commonly used machines are hydraulically powered in-line screw machines, although electric machines are appearing and will be more dominant in the market in near future. The main units of a typical injection moulding machine are the clamping unit, the plasticizing unit, and the drive unit;

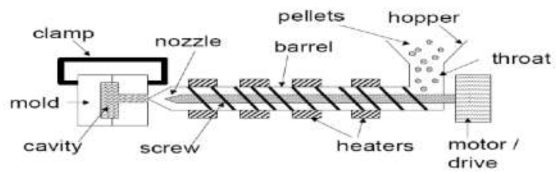

Fig. 3. Injection Moulding Machine

The clamping unit holds the mould. It is capable of closing, clamping, and opening the mould. Its main components are the fixed and moving plates, the tie bars and the mechanism for opening, closing and clamping. The injection unit or plasticizing unit melts the plastic and injects it into the mould. The drive unit provides power to the plasticizing unit and clamping unit. Injection moulding machines are often classified by the maximum clamp force that the machine can generate. This is the force that pushes the two mold halves together to avoid opening of the mould due to internal pressure of the plastic melt in the mould. The clamping force of typical injection moulding machines range from 200 to 100,000 kN.

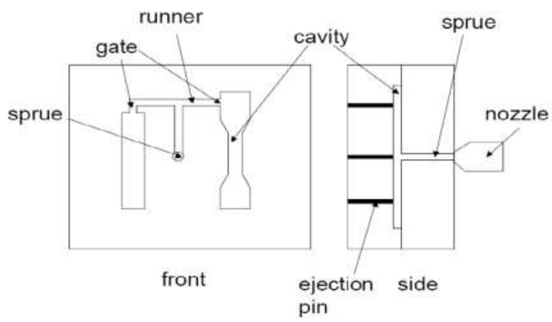

Fig.2 Line Diagram Of Mould

THE INJECTION MOULDING CYCLE

There are three main stages in the injection moulding cycle; stage 1, injection, followed by stage 2, holding pressure and plasticating, and finally, stage 3, ejection of the moulded part. When stage 3 is completed, the mould closes again and the cycle starts over again.

Stage 1 INJECTION OF THE PLASTIC MELTS INTO THE MOULD: In stage 1, the mould is closed and the nozzle of the extruder is pushed against the sprue bushing of the mould. The screw, not rotating at this point, is pushed forward so that the plastic melt in front of the screw is forced into the mould.

Stage 2 HOLDING PRESSURE AND PLASTICATING: When the mould is completely filled, the screw remains stationary for some time to keep the plastic in the mould under pressure, this is called the “hold” time. During the hold time additional melt is injected into the mould to compensate for contraction due to cooling. Later, the gate, which is the narrow entrance into the mould, freezes. At this point the mould is isolated from the injection unit. However, the melt within the mould is still at high pressure. As the melt cools and solidifies, the pressure should be high enough to avoid sink-marks, but low enough to allow easy removal of the parts. During the plastication stage, the material is pushed forward from the feed hopper through the barrel and towards the nozzle by a rotating screw. When the gate freezes, the screw rotation is started. The period of screw rotation is called screw “recovery”. The rotation of the screw causes the plastic to be conveyed forward. As the plastic moves forward, heat from the electric heater bands along the barrel and shear starts to melt the plastic. At the discharge end of the screw, the plastic will be completely melted. The melt that accumulates at the end of the screw pushes the screw backward. Thus the screw rotates and moves backward at the same time. The rate at which plastic melt accumulates in front of the screw can be controlled by the screw backpressure, that is, the hydraulic pressure exerted on the screw. This also controls the melt pressure in front of the screw. When sufficient melt gets accumulated in front of the screw, the rotation of the screw stops. During screw recovery the plastic in the mould is cooled, but typically the cooling is not finished by the end of screw recovery. As a result, the screw will remain stationary for some period until cooling is completed. This period is often referred to as “soak” time. During this time additional plastic will melt in the extruder from conductive heating. Also, the melted material will reach more thermal uniformity, although the soak time is usually too short to improve thermal homogeneity significantly.

Stage 3 EJECTION: When the material in the mould has cooled sufficiently to retain its shape, the mould opens and the parts are ejected from the mould as shown in Fig. 2.2. When the moulded part has been ejected, the mould closes and the cycle starts over again. The different stages can be graphically illustrated as shown in Fig. 2.3. The top bar shows the movement of the extruder screw, the second bar shows the action going on inside the mould and the third bar indicates at what times the mould is open and closed. As can be seen in Fig. 2.3 the major part of the injection moulding cycle is the cooling time required for the plastic in the mould to reduce to a temperature where the part can be removed without significant distortion. The main variable that determines the cooling time is the thickness of the molded part.

THE MOULD Each mould, sometimes referred to as a tool, is built to exact specifications of the part or parts required by the customer. The mould typically consists of two mould halves. Usually one mould half contains the cavity and forms the outer shape of the part. This part of the mould is called the cavity side. The other mould half contains a protruding shape and forms the inner shape of the part, this mould part is called the core. When the core is clamped against the cavity, the hollow space that is formed defines the shape of the part to be moulded. The plastic is usually injected into the mould from the cavity side. The mould cavities are cut to dimensions larger than the desired part dimensions to compensate for the plastic shrinkage which occurs during cooling. The cavity dimensions are equal to the part dimensions plus some shrink factor supplied by the material manufacturer. There are usually two shrink factors given, one for dimensions in the direction of the flow and the other for dimensions perpendicular to the direction of the flow. Estimating shrinkage, however, is not straight forward. It is often difficult to predict the melt flow path in parts with complex geometries and therefore, it is not clear which shrink factor to apply. Part shrinkage is also influenced by the process conditions.

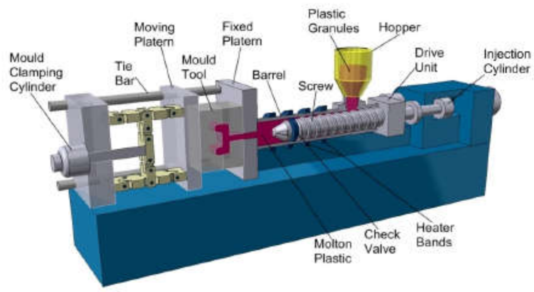

Fig. 4. Injection Molding Machine

The injection molding process

The injection molding process stages starts with the feeding of a polymer through hopper to barrel which is then heated with the sufficient temprature to make it flow , then the molten plastic which was melted will be injected under high pressure into the mold the process is commonly known as Injection, After injection pressure will be applied to both platens of the injection molding machine(moving and fixed platens) inorder to hold the mold tool together afterwards the product is set to cool which helps it in the solidification process. After the product gets its shape the two platens will move away from eachother inorder to separate the mold tool which is known as mold opening and finally the molded product is ejected or removed from the mold. And the process will repeat itself.

DESIGNING PROCEDURE

Starting NX

Toolbars and tools

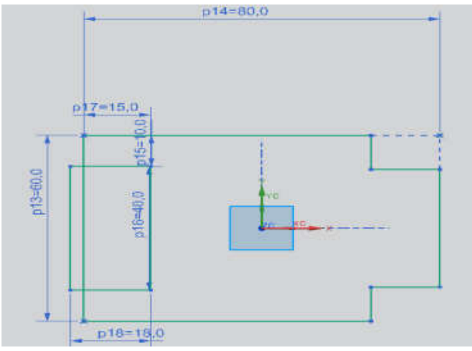

Fig. 5 shows project 2d sketch view

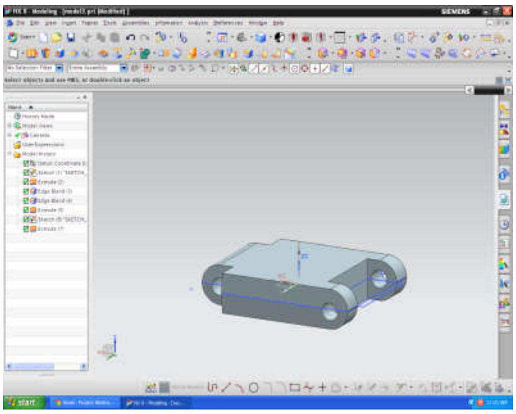

Fig.6 shows the 3d component with model tree

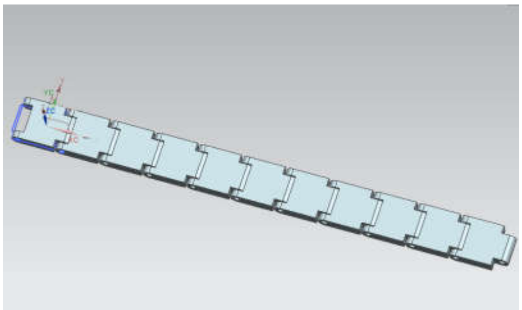

Fig. 7 3d assembly of conveyor chain

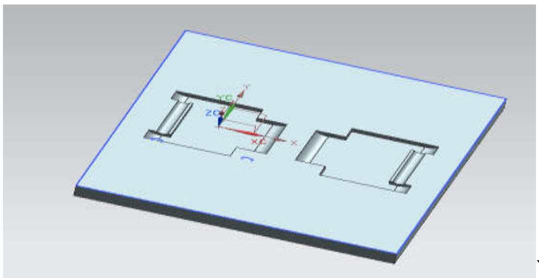

Fig. 8 mold cavity

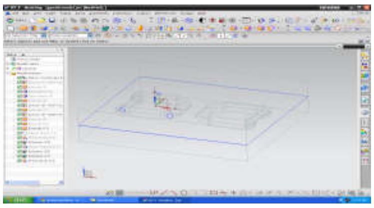

Fig.9 shows the assembly of cavity parts

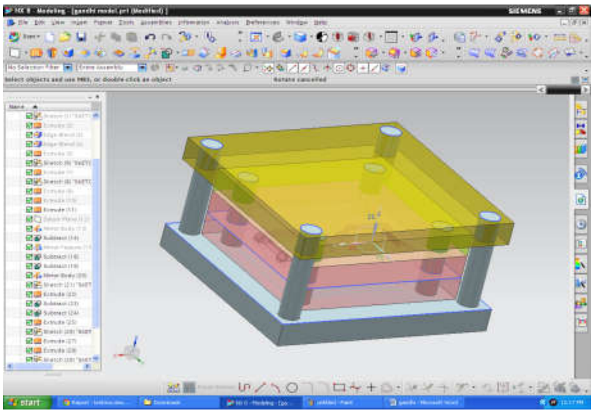

Fig. 10 shows the assembly of Mold base

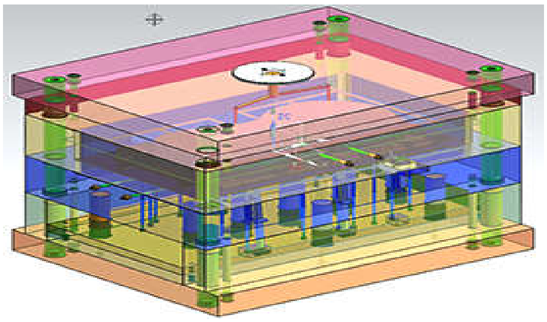

Fig. 11 shows the assembly of Mold with ejection

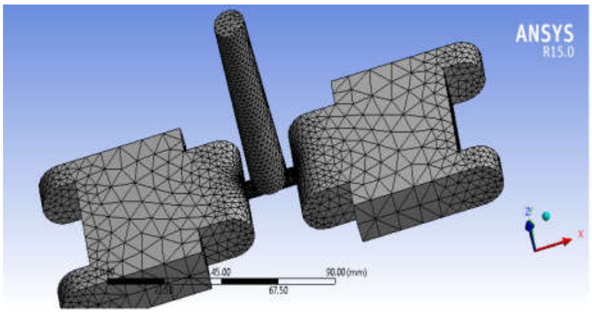

Fig. 12 Meshed model

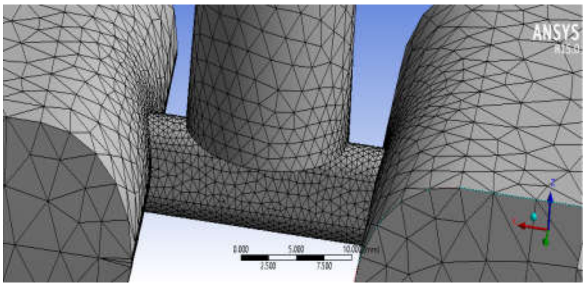

Fig. 13 Zoomed Meshed Model at the Gates

A 3-Dimension tetrahedron mesh element was used for meshing the Mold

BOUNDARY CONDITIONS

Flow inlet and wall has been defined as boundary conditions.

Mass Flow rate of 0.05kg/sec was assigned and no slip stationary wall for the wall.

Inlet temperature of 400K was used.

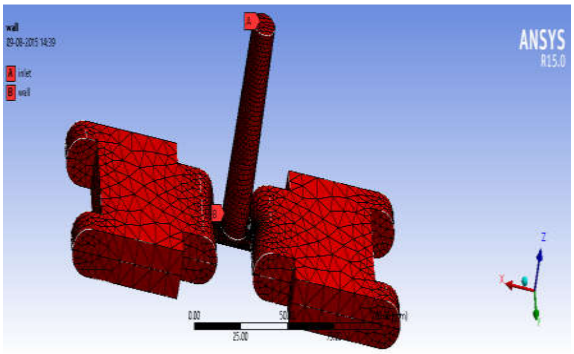

Fig. 14 Boundary Conditions

MATERIAL USED

Domain-Fluid

Propylene(c3h6)

Density 1.7kg/m^3

Specific hear (j/kg-k) Piecewise-polynomial

Thermal Conductivity(w/mK) 0.0168

Viscosity (Kg/m-s) 8.7e-06

3.ANSYS RESULTS AND DISCUSSIONS

Figure shows the velocity vectors across the mold. The velocity was initially decreased and then increased at the mouth of the mold this is due to divergent section of the inlet

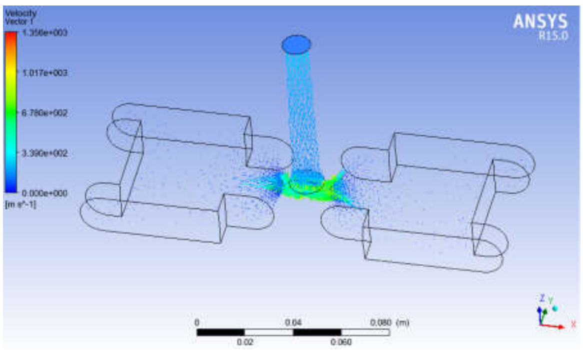

Fig. 15 Velocity Vectors across mold

Figure shows the Velocity stream lines across the mold and fluid rush into the mold cavity.

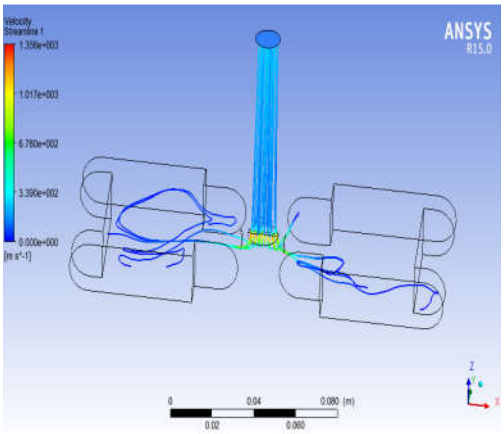

Fig. 16 Velocity Stream Lines

Figure shows the temperature contours across the horizontal plane defined at the Top of the mold, Maximum temperature of 403K was observed at the exit of the inlet and was reduced to 300K at the walls.

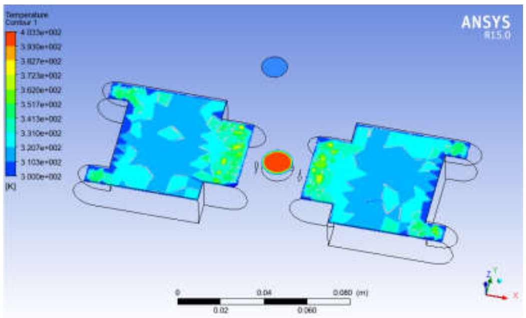

Fig. 17 Temperature Contours in the horizontal plane at the Top section of the mold

Figure shows the variation of temperature across the horizontal plane at the midlle section of the Mold , The area of maximum temperature was found to be more at this section. Maximum temperature as previously said it was found to be 403K.

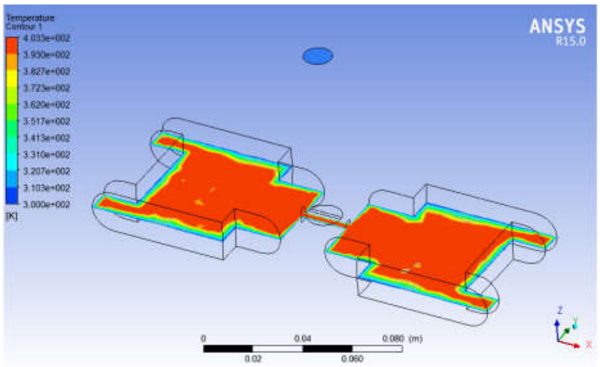

Fig.18 Temperature Contours in the horizontal plane at the Middle section of the Mold

Figure below shows the temperature contours at the horizontal planes For the 3 sections at top bottom and centre of the mold.

It was observed that the Maximum temperature area was more at the center plane then in the bottom plane then in the top plane. This was due to the placing of the inlet exit at the center plane of the mold.

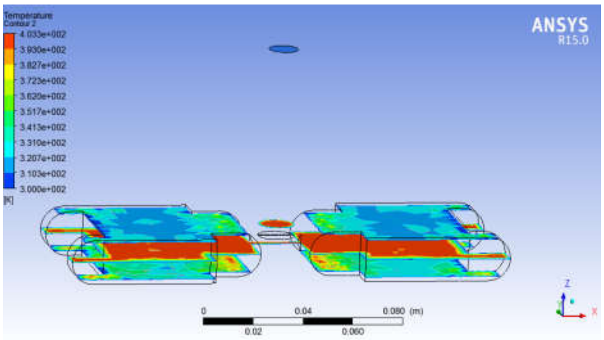

Fig. 19 Temperature Contours in the horizontal plane at the Top middle and bottom section of the Mold

Figure shows the presuure variation at the exit of the inlet, The maximum pressure of injection was found to be 1.792Mpa at the exit of the inlet. A Probe is used to measure the pressure at the point in ansys workbench

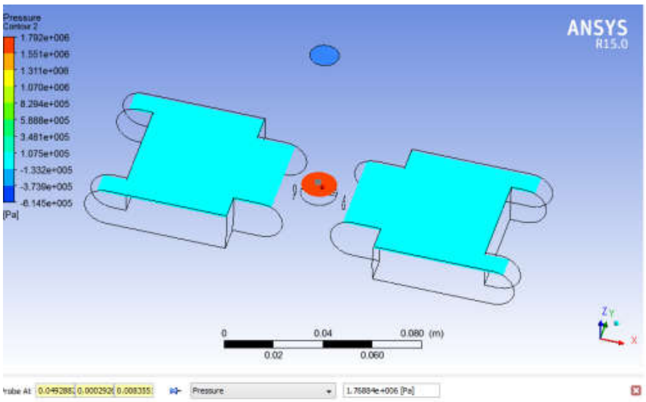

Fig. 20 Pressure Probe at the gate of the Mold

4. RESULTS

Research shows that the integrated using software ANSYS Workbench as injection molds stiffness and strength analysis tools, which can comprehensively reflect the cavity' s force and distortion situation in injection molding process, thus providing scientific and reliable theoretical foundation for mold design and condition for subsequent development of injection mold stiffness and strength analysis system.

5. Conclusions

1. In making the mold it was necessary to have the best possible product design So that it won't complicate the mold designing process.

2. With all the necessary dimensions and by the help of UNI-Graphics product design was achieved. In this phase there were lots of ups and downs in trying to figure out what the best closing system for the wallet might be, a lot of designs were drawn due to the fact that wallet closing system complication complicates the mold design which intern complicates the manufacturing process of the mold, In this thesis it was crucial to find out if there were any defects in the product design and also finding out some important values like material selection, Fill time, Fill pattern and Clamping force.

3. By using the simulation and analysis software Mold flow the above values have been achieved and there were no defects found on the product design.

FUTURE SCOPE

. Further researches can be done on the above designs stage to put the layout of a Cooling system for the mold. The function of the cooling system of a plastic injection mold is to provide thermal regulation in the injection molding process. As the cooling phase generally accounts for about two-thirds of the total cycle time of the injection molding process, efficient cooling is very important to the productivity of the process.

References

[1] Georg menges , Hans Pecker "Computer Aided Engineering in Injection molding" Journal on comprehensive polymer science & Supplements, vol .7 1989 .Pages 451-488.

[2] Millers and Pedgen D, 2000, "Introduction to Manufacturing Simulation", Proceedings of the 2000 winter simulation conference PP 63-66.

[3] A.H.S.Park ,X.P.Dang, A.Roderburg, "Development of Plastic Panels " CIRP Journal of Manufacturing &Techonology vol 26 Pages 35-53.

[4] Alan.R.Grayer, "Alternative approaches in Geometric modeling", Journal on CAD, volume 12, issue 4,july 1980, pages 189-192.

[5] Erh Guoliangxu, "Mixed Finite Element Methods for Geometric modeling using 4th order geometric flows. "Journal on Computer Aided Geometric Design". vol 26, issue 4, may 2009,Pages 378-395.

[6] E.G.Gottstain, M.Goerdeler, G.V.S.S Prasad "Mechanical Properties :Plastic Behaviour" Journal on Encyclopedia of condensed matter of Physics"2005 Pages 298-305.

[7] C.Khan molek. L.Robert,A.Singh . **"**High Resolution Thermo plastic Rapid Manufacturing using Injection Moulding**".** CIRP Journal of Manufacturing science and technology**,** volume 4, issue 4 2011, Pages 382-390.

[8] E.Maine, M.F.Ash by **"**Material selection & Mechanical Design **"**Journal on Encyclopedia of Material ,Science &Technology 2nd edition pages 53-83.

[9] M.zakria Baig, **"**Industrial Engineering Estimating &Costing**",** Radiant Publishing House

[10] Japan.s.Daniel.L. and Theodor .k.2005. **"**Finite Element Analysis of Beams**"**, Journal of Impact engg**.** vol 31, Pages 861-876.

[11] Peacock AJ (2000) Handbook of polyethylene. Marcel Dekker: 13–30.

[12] Elias, Hans-Georg. (2003).An Introduction to Plastics. Weinheim: Wiley-VCH GmbH & Co. KGaA.

[13] Antunes, M.; Velasco, J.I.; Realinho, V. & Solorzano E. (2009). Study of the cellular structure heterogeneity and anisotropy of polypropylene and polypropylene nanocomposite foams. Polymer Engineering and Science. Vol.49, No.12, pp. 2400-2413, ISSN 1548-2634.

[14] S Huang, H.-X.; Wang, J.-K. & Sun, X.-H. (2008). Improving of Cell Structure of Microcellular Foams Based on Polypropylene/High-density Polyethylene Blends. Journal of Cellular Plastics, Vol.44, No.1, pp. 69-85, ISSN 1530-7999.

[15] S R.T. Moura, A.H. Clausen, E. Fagerholt, and M. Alves. Impact on HDPE and PVC plates – Experimental andnumerical simulations. Int. J. of Imp. Engng., 2010. in press.

[16] Lei, Xie ; Gerhard , Ziegmann . (2009) Influence of processing parameters on micro injection molded weld line mechanical properties of polypropylene ,Microsystem Technology, 15, pp 1427–1435 .

[17] Lee ,K.S; Lin, J.C. (2006) Design of the runner and gating system parameters for a multi-cavity injection mould using FEM and neural network, , International Journal of Advanced Manufacturing Technology , vol.27,pp. 1089–1096.

[18] Raviwongse ,Rawin; Allada., Venkat; (1997) Artificial Neural Network Based Model for Computation of Injection Mould Complexity ,International Journal of Advanced Manufacturing Technology , 13, pp. 577-586.

[19] Jan, T. C. and O'Brien, K. T. (1993) A user-friendly, interactive expert system for the injection moulding of engineering thermoplastics.International Journal of Advanced Manufacturing Technology, **8**, 42–51.

[20] Chen, B. S. and Liu, W. H., "Numerical simulation and experimental investigation of injection mold filling with melt solidification", Polymer Eng. and Sci. **29** (1989) 1039–1050.

[21] Pandelidis, I. O. and Kao, J.-F. (1990) DETECTOR: a knowledge-based system for injection molding diagnostics. Journal of Intelligent Manufacturing, **1**, 49–58.

[22] Shi,F; Lou,Z. L.; Lu ,J. G. and Zhang Y. Q. (2003), Optimization of Plastic Injection Molding Process with Soft Computing operations ,International Journal of Advanced Manufacturing Technology ,21,pp.656–66.

[23] P. Postawa, J. Koszkul, Influence of processing conditions on changing of shrinkage and mass POM injection molding parts, Proceedings of the 13th Scientific International Conference, Achievements in Mechanical and Materials Engineering, AMME'2005, Gliwice-Wisáa, 2005, 531-534.

[24] Mathivanan D. ; Parthasarathy, N. S. (2009) .Prediction of sink depths using nonlinear modeling of injection molding variables, International Journal of Advanced Manufacturing Technology, vol. 43 pp.654–663.