IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

FINITE ELEMENT ANALYSIS AND DESIGN OPTIMIZATION OF PLATEN FOR INJECTION MOLDING MACHINE

[1] Jay R. Modi, [2] Prof. Dhaval A. Patel

[1] M.Tech Student, [2] Associate Professor,

[1] Department of Mechanical Engineering,

[1] Sankalchand Patel College of Engineering, Visnagar, Gujarat, India

Abstract: Plastics are certainly most versatile of all known materials today and have therefore, entrenched themselves in enviable position from where are not even possible to replaced. Injection molding machine is one of the most widely used method of conversion of plastic into various end products application to wide range of plastic materials from plastic commodity to specialty engineering plastic. In injection molding machine, platen plays a predominant role. During the process generally compressive stress generates at particular regions. As load varies with fast rate there is chance to failure of tie bar rod. Due to heavy mould shape, size of platen also changes, that's increase its weight as well as stress level at certain region and this is not good in practice. As this leads to failure of platen or failure of tie rod due to stretching by nut and platen. The aim of this research is study about those areas where stress can affect the failure of tie bar due to heavy weight of platen the deflection or misalignment with movable platen. Finally redistribution of stress so that uniform stress is achieved (optimization of platen). This project is including Finite Element Analysis and Optimization of a Typical Structural Component i.e. stationary platen of a Plastic Injection Molding Machine. The aim of project is to optimize a typical structural component (platen) by using finite element analysis after checking induced stresses with allowable design stress. Optimize design of platen with the use of removing material from them. Hence design modification of platen is carried out to achieve good strength and cost effectiveness.

***Index Terms* – Injection Molding Machine, Stationary Platen, Plastics, FEA, Design Optimization.**

[1] INTRODUCTION

Plastics are certainly the most versatile of all known material today and have, therefore, established themselves in an enviable position from where they are not even possible to be displaced. The manufacturing of injection molding machines is one of the world's fastest growing industries in accompany with the application of plastic production. The development of precision injection molding techniques has attracted more and more attention in the world. Precision injection molding techniques include injection molding process and injection molding machines.

The most common method of conversion of plastics are : -

- Compression Molding
- Injection Molding
- Blow Molding
- Extrusion and Thermoforming

Injection molding is one of the principle methods of conversions of plastics into various end-products using a very wide range of plastics materials from commodity plastics to specialty engineering plastics.

A large variety of molding machines are manufacture in the country with the indigenously developed technology as well as in collaboration with the world leaders to indigenously manufacture machines of world standards. Clamping force, low pressure molding machines gas injection molding machines, multi component molding machine's co-injection molding machines or some very special tailor-made machines dedicated to specific end-uses and polymers.

The facilities for mold making have also developed over the years and even the most difficult and sophisticated molds are now being designed and fabricated in India. The potential for the injection molded products is tremendous in our country

[2] COMPONENTS OF INJECTION MOLDING MACHINE

Injection Molding machines are basically divided in two units

- Injection Unit
- Clamping Unit

Further, Injection Unit comprises of Barrel &Screw Assembly, Injection Cylinder, Thrust Housing, Drive Unit, Hopper, Heaters, Ring Plunger, Nozzle.

And Clamping Unit incorporates Moving and Stationary Platens, Tie Bars, Mold, Mold Clamping Cylinder, Toggle Links, Ejector, Nuts.

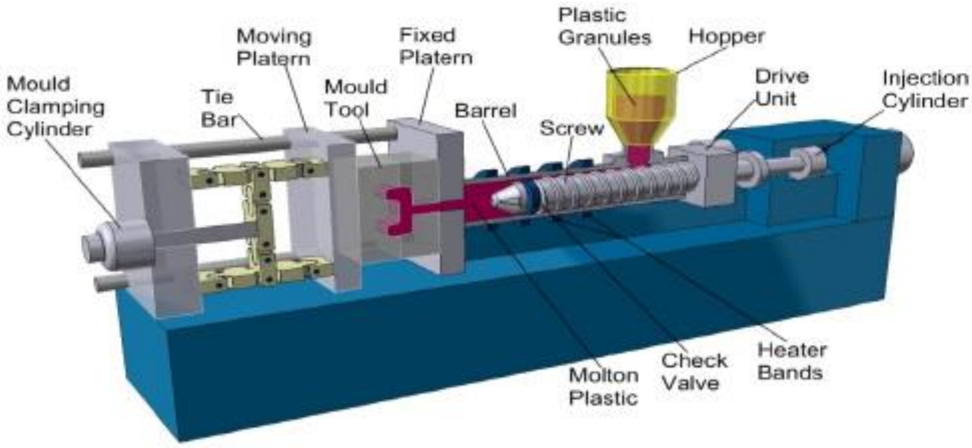

Figure 1- Injection Molding Machine

[3] WORKING PRINCIPLE OF INJECTION MOLDING MACHINE

The main working principle of injection molding machine is to use the plastic thermal physical properties. Firstly, loader loads the plastic material into the barrel, which is surrounded by heaters melting the plastic in the barrel which is assembled with servo motor screw. The plastic melt will flow up the screw and under the heating condition get tight and move forward to the screw head. At the same time, due to the plastic reacting force, the screw will step back. So at the screw head, it formed a plastic melt saving space to finish its plasticizing process. At the same time, under the injection hydraulic cylinder force, the screws will inject the plasticizing melt into the plastic mould through the nozzle. The plastic melt will remain in the mould cavity through the holding pressure, cooling, process, and then formed into solid shape and final product is ejected out of the mould. So in injection molding machine main recycle process including : quantitative feeding – plasticizing – injection - molding the shape-pick up the piece, then close the mould to the 2nd recycle process

Clamping → Pre-molding → Reverse → Nozzle advance → Injection → Hold pressure → Nozzle retreat → Cooling → Open mold → Eject → Open door → Take workpiece → Close door → Close mold

[4] LITERATURE REVIEW

C. Sasikumar et al. In this paper analyses premature failure of tie bars, the tie bars are subjected to a pulsating cyclic loading during the injection molding process cycle. The failure occurred at the root of the first thread of transverse fatigue fracture induced by a pulsating tensile stress with multiple points of high stress concentration. The high stress concentrations appear to have introduced with an amalgamation of indecent molding parameters resulting in uneven tensions in the four tie bars and aggravated by the presence of some material defects. The material defects observed are inclusions, presence of some retained austenite and fine cracks. The tie bar of the injection-molding machine has failed at the root of the thread by fatigue fracture induced in a combination of pulsating cyclic tensile stress and a component of torsional stress with multiple points of high stress concentration. High stress concentrations have introduced with a combination of improper molding parameters resulting in uneven tensions in the four tie bars and significant amount of material defects. The fatigue crack has initiated at the root of the final thread at an inclusion and propagated through the inclusions and fine cracks inherently present in the material. Once the diameter of the tie bar is reduced due to the propagation of fatigue fracture and reaches the critical diameter, the final failure has occurred by ductile fracture due to overload.

George Z. Voyiadjis et al In this paper researcher presented a non-linear FE analysis, for the elasto-plastic behavior of thick shells and plates including the effect of large rotations. In the treatment of material non-linearities the authors adopt:

A non-layered approach and a plastic node method.

A yield function expressed in terms of stress resultants and stress couples modified to investigate the development of plastic deformations across the thickness, as well as the influence of the transverse shear forces on plastic behavior of plates and shells.

Isotropic and kinematic hardening rules.

Hence, Non-linear finite element analysis had used for assembly analysis to analyze the exact values of stress and deflection at the moving platen because there is a bonded contact between the mating components of assembly, therefore there will be a non-linear behavior of the analysis.

Patel Niral et al. The paper consist the detail design of clamp cylinder for 1000 ton injection machine with topology optimization. The design is carried out based on calculated diameter and thickness. Modeling and FEA is doing for newly designed 1000T clamp cylinder and to be verified with theoretical calculation and acceptance criteria. The topology optimization of clamp cylinder is also carried out using CAE tools to reduce weight with the constraints of standard operating condition. Topology optimization is a mathematical approach which optimizes the material layout within a given design space, under given set of loading and boundary condition. Fourteen iterations were required to solve the optimization problem. The optimized model is equally strong and light in weight compared to existing model. After modify the topology of the clamp cylinder the maximum principle stresses and the total deformation within the permissible limits, but substantial reduction of the material hence weight have been obtained. Hence the purpose of the topology optimization serves. The topology optimization of the component is carried out and substantial reduction in weight about 70 kg is obtained.

Dheeraj Mandliya et al. In this paper researcher to study about those areas where stress can affect the failure of tie bar due to heavy weight of stationary platen the deflection or misalignment with movable platen.

During the process generally compressive stress generates at particular regions. As load varies with fast rate there is chance to failure of tie bar rod. Due to heavy mold shape, sizes of platen also change, that's increase its weight as well as stress level at certain region and this is not good in practice. This leads to failure of platen or failure of tie rod due to stretching by nut and platen. This paper is including Finite Element Analysis and Design Optimization of a Typical Structural Component of a Plastic Injection Molding Machine. The aim of paper is to optimize a typical structural component (stationary platen) by using finite element analysis after checking induced stresses with allowable design stress. FE analysis of existing stationary platen is carried out by using ANSYS software. Finally In optimization design modification has been carried out in Pro/E model and checked for its feasibility with respect to stresses and weight. The aim of the optimization is reduce the weight and make it cost effective. Existing model of stationary platen has dome type shape. Dome type shape is converted in to box type which resulted in reduction of overall thickness of platen to the tune of 5 %.

Shu Huang Sun et al. In this paper researcher main objective is to design the structure of the stationary platen with optimized stiffness at minimal raw material cost. Tie bars are key components of a plastic injection machine. They very easily fatigue in periodically long term operations due to the bending moment transferred to them by the bending of the stationary platen. This problem can be easily overcome by reducing the deflection of the stationary platen through topology optimization of the platen structure by applying a cost or weight constraint. In this paper, the self-organization method was introduced to optimize the topology of the stationary platen. This method is to modify the Young's modulus of each element according to the ratio of its stress and the average stress of the entire model after each FEM analysis. By applying this method to the stationary platen design, the deflection of the platen could be reduced, which correspondingly reduces the bending load of the tie bars and thus extending their operating life. The proposed stationary platen design was almost identical to that used in a commercial machine Also, the proposed method herein, proved the effectiveness of the existing platen design through a simple and systematic scientific way.

[5] METHODOLOGY

A. Basic Assumptions of the Force Conditions of the Stationary Platen

Topology optimization design of the stationary platen can be carried out in a 250 ton precision plastic injection molding machine with clamp force 2451kN. The material of the stationary platen is SG 450/10 IS1865, where the Young's modulus of each element is set to 1.69 and the Poisson's ratio is 0.275.

In order to save computing resources and give prominence to the influence of forming precision, we have made the following basic assumptions of the force conditions of the stationary platen.

- The stationary platen is subjected to the maximum bending during the injection and holding-pressure stages. We could analyze and optimize the stationary plate during the injection and holding-pressure stages to ensure the stiffness and strength of the stationary platen.
- Four tie bars have the movement guide function for the stationary platen, and they have less effect on the deformation of stationary platen during the holding pressure stages. The structural topology optimization mainly involves the calculations on the stiffness and strength of the stationary platen, so the influence of tie bars can be negligible.
- During the injection and holding-pressure stages, the internal pressure of the clamping cylinder is 16 MPa. Considering the influence of forming accuracy, the pressure of clamping cylinder could be converted to Z displacement constraints at the wall of clamping cylinder during the optimization process.
- The stationary platen is fixed to the frame. Compared with the clamping force of 1600 kN, the gravity of the stationary platen can be negligible.
- We take the structure of the stationary platen as a cast iron. The screw holes and other small holes can be negligible, as to simplify the structures of the stationary platen.

B. Material Properties

- Spheroidal Gray Cast Iron is the material used for Injection Molding Platen in all Industries
- It's chemical name is SG 450/10 IS1865

Tensile strength, N/mm^2	450
Elongation, %	10
Density, kg $/mm^3$	$7.1x\ 10^{-6}$
Young's Modulus, MPa	169000
Poisson's Ratio	0.275
Bulk Modulus, MPa	125190
Shear Modulus, MPa	66275

Table 1 - Material Properties

C. Stationary Platen 3D Model – Iteration-I

- Stationary Platen used in successfully running Injection Molding machine of 250 tonnes is taken for optimization and CAD model for the same generated using PTC CREO
- Platen Size is 1090 x 1050 x 245 mm & it is weighted 1137.80 kg
- Force = 250 tons ~ 2451700 N is to be subjected on to the stationary platen

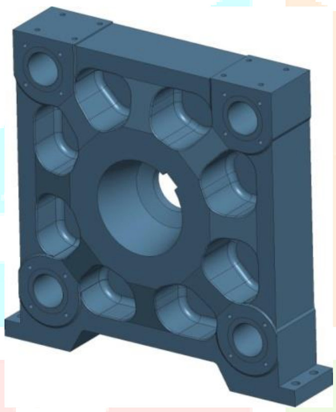

Figure 2 - Stationary Platen 3D model – Iteration-I

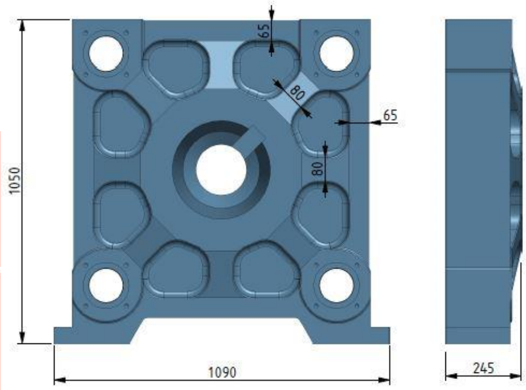

Figure 3 - Stationary Platen Drawing – Iteration-I

D. Stationary Platen 3D Model – Iteration-II

- Iteration-II was created after certain permissible modification into the existing design which included thickening of supporting members leading to reducing the pocket sizes
- Though it weight raises to 1188.40 kg

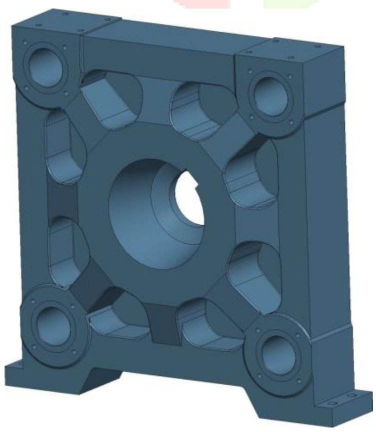

Figure 4- Stationary Platen 3D model – Iteration-II

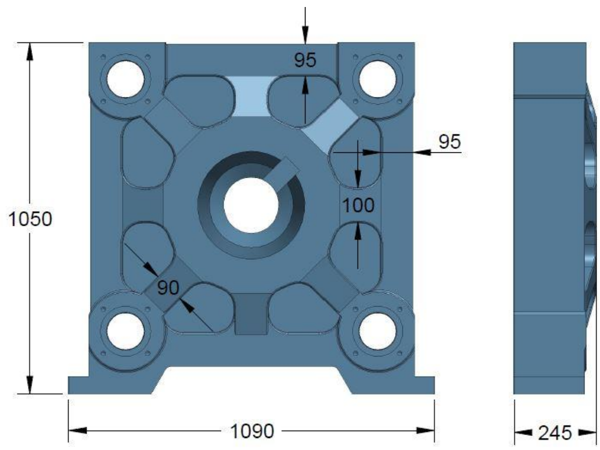

Figure 5 - Stationary Platen Drawing – Iteration-II

E. Stationary Platen 3D Model – Iteration-III

- Further Iteration 02 was created after certain permissible modification into the existing design which included increasing the pocket sizes and reducing overall weight of it to 1069.5 kg

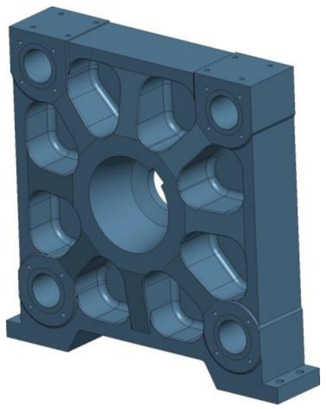

Figure 6 - Stationary Platen 3D model – Iteration-III

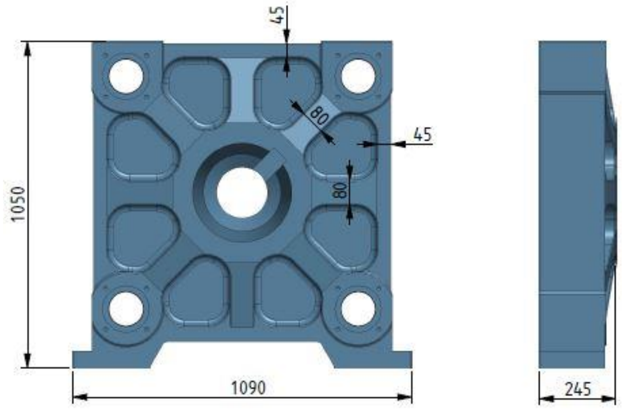

Figure 7 - Stationary Platen Drawing – Iteration-III

F. FE Analysis of Platen

- Static FE analysis has been carried out on Platen for checking stress and weight. Here, all four tie bar and nut mounting face will be considered as a fixed supports. And the force direction will be considered from the mold mounting face of platen shown by red arrow, where force will be considered as 250 tons for analysis purpose.
- Equivalent Stress, Maximum Shear Stress, Total Deformation of all iteration will be analyzed for finalizing conclusion.
- The below figure show geometry of stationary platen and then generate mesh on it.

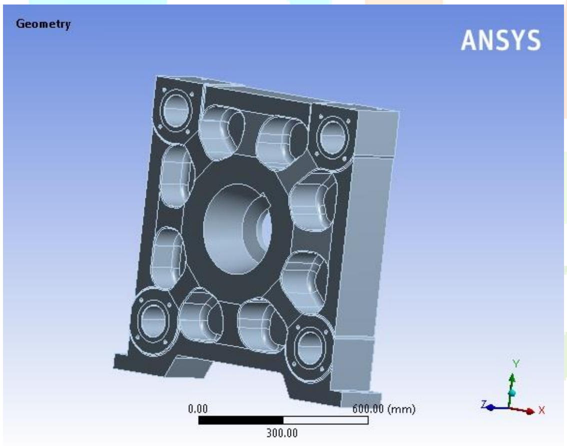

Figure 8 - Geometry of Stationary Platen

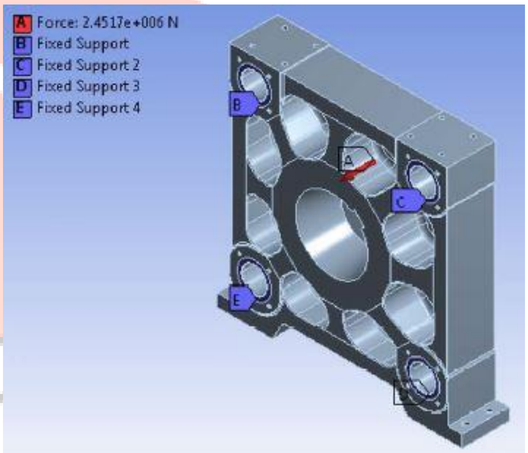

Figure 9 - Static Structural Analysis providing fixed support and Forces

G. Thermal Analysis of Stationary Platen

.

- In stationary platen, nozzle and injection unit is mounted on center for injection for mold in clamping unit.
- Considering thermal analysis on center of platen. As, Nozzle get heated during injection stroke in injection unit thus, this temperature may effect on stationary platen and it get deformed. Here temperature applies on center of platen for analysis of thermal effect.

Total deformation $\delta l = \alpha * l * \delta t$

Where, δl = total deformation
α = coefficient of thermal expansion = constant
$\delta t = (t_{max} - t_{min})$

On substituting all values in equation we get,
Total deformation δl = 0.07 mm

- Hence, it is very low value and thus it can be negligible.

[6] RESULTS AND DISCUSSION

	Unit	Iteration- I	Iteration-II	Iteration-III
Density	kg /mm^3	0.0000071		
Young's Modulus	MPa	169000		
Poisson's Ratio		0.275		
Bulk Modulus	MPa	125190		
Shear Modulus	MPa	66275		
Width	mm	1090		
Height	mm	1050		
Thickness	mm	245		
Volume	mm^3	160250000	167380000	150630000
Mass	kg	1137.80	1188.40	1069.50
Force	N	2451700		
Equivalent Stress (Min)	MPa	0.0041162	0.0034258	0.0034347
Equivalent Stress (Max)	MPa	718.30	667.44	858.20
Equivalent Stress (Avg)	MPa	28.54	32.26	30.529
Maximum Shear Stress (Min)	MPa	0.0021456	0.0018398	0.0019597
Maximum Shear Stress (Max)	MPa	393.37	365.67	471.71
Maximum Shear Stress (Avg)	MPa	15.355	17.53	16.41
Total Deformation (Min)	mm	0.00	0.00	0.00
Total Deformation (Max)	mm	0.27044	0.26038	0.30204
Total Deformation (Avg)	mm	0.071643	0.090123	0.081049

Table 2 - FEA Result Comparison

Table shows comparison of results obtained after analyzing all different iterations of stationary platen in ANSYS software. Firstly all the Iterations are subject to same conditions and are subjected to 250 tonnage of force. Overall dimension of platen is 1090 X 1050 X 245, though minor modifications in design results in reduction of weight marginally. By comparing each result of all three iterations we can easily summarized that iteration I weighing 1137.80 kg has the least deformation i.e. 0.07 comparing to Iteration II & III which tends to deform up to 0.09 & 0.08 respectively

[7] CONCLUSION

In all this research, many parameters were analyzed for material removal from varied components of injection molding machine within the permissible limit of stresses, deformation, linear elastic fracture mechanics analysis of platen are carried out. Optimization has been carried out with help of variety of method and the best factor settings are considered in respective cases.

Optimization of all responses has been carried out and the best factor setting obtained is Stationary Platen weighing 1137.80 kg. This Iteration-I gives optimum value of material removal from platen. Hence, with use of this iteration we have optimized 6% weight of total weight of platen.

REFERENCES

[1] Sasikumar C. and Srikanth S. and Das SK, “Analysis of premature failure of a tie bar in an Injection Molding Machine”, Engineering Failure Analysis, 13 (8). pp. 1246-1259, Jan 2006. ISSN:2077-2455

[2]George Z. Voyiadjis and Pawel Woelke: General non-linear Finite Element Analysis of Thick Plates and Shells, International journal of solids and structures Vol. 32. 30th August 2005. ISSN:0020-7683

[3] Patel Niral, Mihir Chauhan “FEA and Topology Optimization of 1000T Clamp Cylinder for Injection Molding Machine” Nirma University International Conference (NUiCONE 2012), Procedia Engineering 51 (2013)617-623. ISSN:1877-7058

[4] DheerajMandliya, YogeshAgrawal, G.V.R SeshagiriRao “Design Optimization of Stationary Platen of Plastic Injection Molding Machine Using FEA”, International Journal Of Engineering Sciences & Research Technology, May-2014. ISSN:2277-9655

[5] Shu Huang Sun “Optimum Topology Design for the Stationary Platen of a Plastic Injection Machine”, Computers in Industry, Volume 55, Issue 2, Pages: 147 –158, Oct 2004. ISSN:0166-3615