IJIRST –International Journal for Innovative Research in Science & Technology| Volume 1 | Issue 6 | November 2014
ISSN (online): 2349-6010

Finite Element Analysis & Optimization of Platen of Injection Molding Machine- A Review

Rahul M. Thaker
P G Student
Department of Mechanical Engineering (CAD/CAM)
L.J Institute of Engineering and Technology, Ahmedabad

Ms. Neha B. Joshi
Assistant professor
Department of Mechanical Engineering
L.J Institute of Engineering and Technology, Ahmedabad

Abstract

Plastics are certainly most versatile of all known materials today and have therefore, established 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 stationary platen play a very important 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 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. The aim of this paper is 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.
Keywords: tie bar, clamp cylinder, stationery platen, moving platen, finite element analysis, optimization.

I. 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. 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. The products range covers from simple house wares to very specialized products from electronics, telecommunication, automobile and defence industrial segments. The machines in use are from inexpensive manually operated to the very expensive state-of-art machines which produce components in quality that matches the best in the world thereby opening-up the exports market. A large variety of molding machines are manufactured 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.

The most common methods of conversion of plastics are: -

- Compression Molding
- Injection Molding
- Blow Molding
- Extrusion And Thermoforming

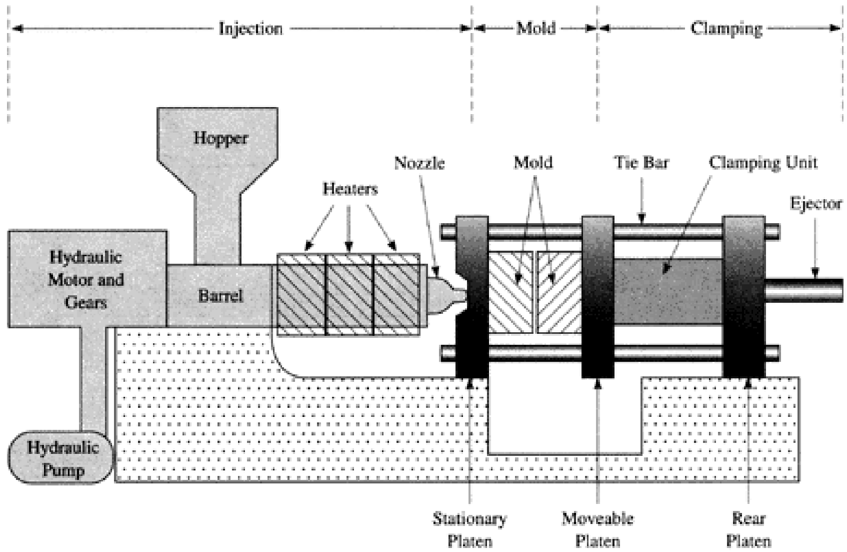

Fig.1: Plastic Injection Molding Machine

The injection system consists of a hopper, a reciprocating screw and barrel assembly, and an injection nozzle, as shown in Fig.2 This system confines and transports the plastic as it progresses through the feeding, compressing, degassing, melting, injection, and packing stages. Thermoplastic material is supplied to molders in the form of small pellets. The hopper on the injection molding machine holds these pellets. The pellets are gravity-fed from the hopper through the hopper throat into the barrel and screw assembly.

Application of Plastic Injection Molding Machine:

- Automotive Structural Parts
- TV Cabinets
- Computer Monitor-housing
- Rigid Packaging Containers

II. LITERATURE REVIEW ON PLATEN OF INJECTION MOLDING MACHINE

A. *Analysis of premature failure of a tie bar in an injection molding machine, Sasikumar, C and Srikanth, S and Das, , Engineering Failure Analysis, 13 (8). pp. 1246-1259, Jan 2006*

In this paper researcher analysis that premature failure of a tie bar in an injection molding machine. The tie bars are subjected to a pulsating cyclic loading during the plastic molding process. The failure was found to occur at the root of the first thread by transverse fatigue fracture induced by a pulsating tensile stress with multiple points of high stress concentration. The high stress concentrations appear to have been introduced by a combination of improper 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. A hydraulic clamping mechanism rather than a toggle clamp mechanism for the mould will minimize the cyclic strain on the tie rods. The tie bar of the injection-molding machine has failed at the root of the thread by fatigue fracture induced by a combination of pulsating cyclic tensile stress and a component of torsional stress with multiple points of high stress concentration. High stress concentrations have been introduced by a combination of improper molding parameters resulting in uneven tensions in the four tie bars and significant amount of material defects. The uneven load has resulted in a torsional component of stress, misalignment of thread and consequent gouging at the threads

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.

B. *FEA and Topology Optimization of 1000T Clamp Cylinder for Injection Molding Machine, Patel Niral , Mihir Chauhan, Nirma University International Conference (NUiCONE 2012), Procedia Engineering 51 (2013)617-623*

The paper consist the detail design of clamp cylinder for 1000 ton injection molding machine with topology optimization. The design is carried out based on calculated diameter and thickness. Modeling and FEA is carried out 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. For the purpose of optimization, clamp cylinder material is taken homogenous, iso-tropic, linear and temperature independent. The topology optimization of the component is carried out and substantial reduction in weight about 70 kg is obtained.

C. Design Optimization of Stationary Platen of Plastic Injection Molding Machine Using FEA, Dheeraj Mandliya, Yogesh Agrawal, G.V.R Seshagiri Rao, International Journal Of Engineering Sciences & Research Technology,May-2014.

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. In injection molding machine stationary platen play a very important 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 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. Hence design modification of platen is carried out to achieve good strength and cost effectiveness. 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 %.

D. Optimum topology design for the stationary platen of a plastic injection machine, Shu Huang Sun, Computers in Industry, Volume 55, Issue 2, Pages: 147 –158, Oct 2004.

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. 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.

E. General non-linear finite element analysis of thick plates and shells, George Z. Voyiadjis and Pawel Woelke, International journal of solids and structures Vol. 32. 30th August 2005.

In this paper researcher presented a non-linear finite element analysis, for the elasto-plastic behavior of thick shells and plates including the effect of large rotations. The small strain geometric non-linearities are taken into account by means of the updated Lagrangian method. 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 has been 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.

III. CONCLUSION

As per review of research papers, we can see that during the process 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 change , that's increase its weight as well as stress level at certain region. This leads to failure of platen or failure of tie rod due to stretching by nut and platen. The aim of this paper is study is 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.

REFERENCES

[1] Sasikumar, C and Srikanth, S and Das, S K, "Analysis of premature failure of a tie bar in an injection molding machine", Engineering Failure Analysis, 13 (8). pp. 1246-1259, Jan 2006.

[2] 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.

[3] Dheeraj Mandliya, Yogesh Agrawal, G.V.R Seshagiri Rao "Design Optimization of Stationary Platen of Plastic Injection Molding Machine Using FEA", International Journal Of Engineering Sciences & Research Technology,May-2014.

[4] 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.
[5] 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.