ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Design Optimization of Stationary Platen of Plastic Injection Molding Machine Using FEA

Dheeraj Mandliya*[1], Yogesh Agrawal[2], G.V.R Seshagiri Rao[3]
*1,2,3 BM College of Technology, Indore, India
dheerajmandliya96@gmail.com

Abstract

Plastic 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. This create loss of money (Production stop), man (injured due to accident), and material (increase inventory cost). The aim of this project 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. Finally redistribution of stress sothat uniforme stress achive (optimization of platen). This project is including Finite Element Analysis and Design Optimization of a Typical Structural Component of a Plastic Injection Molding Machine. The aim of project 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.

Keywords: FEA, Molding machine, Stationary platen, Optimization.

Introduction

An injection molding machine can be defined as a machine that produces formed objects in a discontinuous manner primarily from macromolecular materials.

- The forming is a primary forming under pressure.
- Part of plasticized material in the machine is injected through a channel in to mold cavity.
- Injection molding machines are Composed of three parts:
  (1) The injection unit,
  Which feeds Plastic melt to a
  (2) Mold,
  This is held closed by
  (3) Clamping unit.
- The pressure inside the mold cavity generates a mold opening force which is transferred through shear and compressive stresses to the Centre of the machine platens.
- The platens then transfer the load through shear stresses to the tie bar retainers.
- The retainers, in turn, transfer the load through shear stresses to the tie bars, which should be in pure tension. It should be noted that the mold's guide bushings and the machine tie bars are also subjected to possible lateral loadings.
- This machine design has theoretical minimum number of 20 structural components with 8 axial or compressive stresses and 22 shear stresses.

Methodology

Simplified FE Model

Fig 2.1.Simplified FE Model

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

Quarter Part of Simplified FE Model

- Using Symmetry of Simplified FE model
- So it is converted in to quarter part.

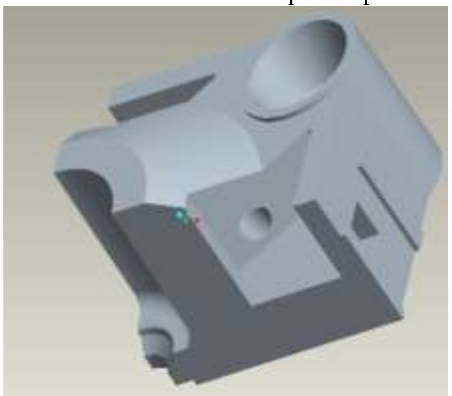

Fig 2.2.Quarter Part of Simplified FE Model

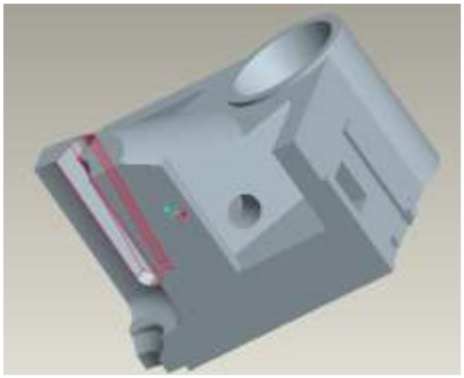

Fig 2.3.Quarter part of FE model with Nozzle And T-slot

Forces Applied On Stationary Platen

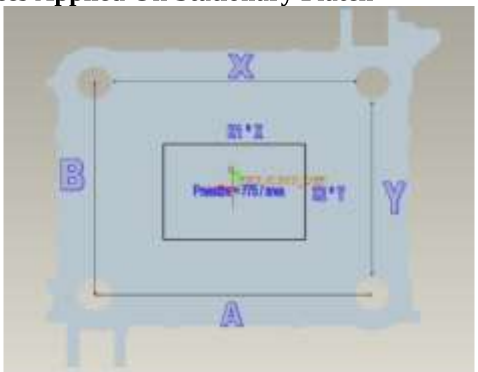

Fig.2.4. Pressure Applied at Projected Area (Back View)

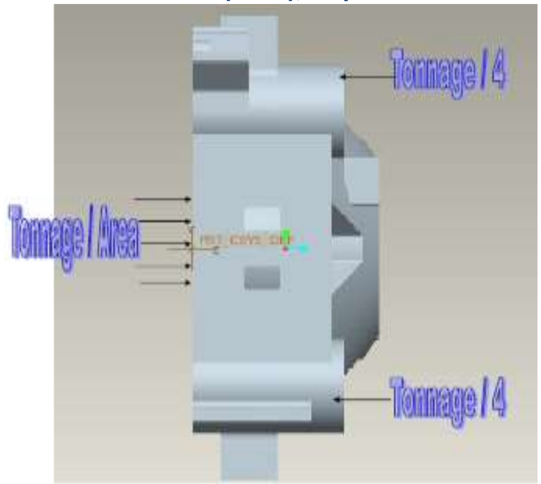

Fig 2.5.Applied forces at each strain rod (Side View)

Data Reduction

- A = 1340 mm
- B = 1140 mm
- X = A – Strain rod diameter
  = 1340 – 190
  = 1150 mm
- Y = B – Strain rod diameter
  = 1140 – 190
  = 950 mm
- Force applied = 775 tonne
  = 775*1000 kg
  = 775*1000*9.81 N
  = 7602750 N
- Pressure applied in the projected-rectangular area = 1.565 kg / mm^2
  = 1.565*9.81 N
  = 15.35 N
- Force applied due to Each Strain Rod = Force / 4
  = 7602750 / 4 N
  = 1900687.5 N
- Maximum bending stress (σ) = 35.68 N / mm^2
- The stress concentration K_1 and K_2 are constants.
- Volume = 1.9863855e+08 mm^3.
- Surface area = 4.6468334e+06 mm^2.
- Density =7.1000000e-09 tonne per mm^3.
- Mass = 1.4103337e+00 Tonne.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

Design Optimization

Design Modification for Optimization

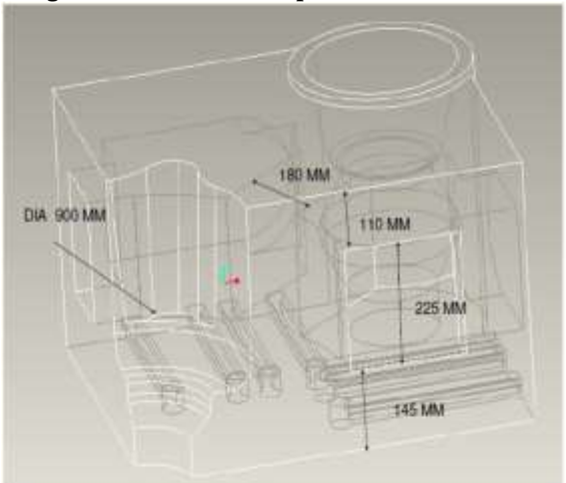

Fig 3.1.Iteration (1) Mass Reduced = 109.28 Kg

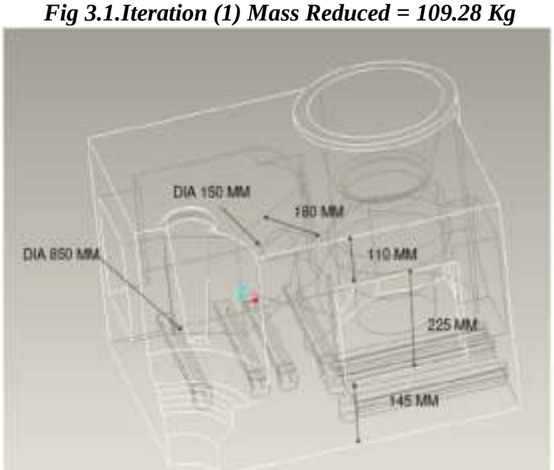

Fig 3.2.Iteration (2) Mass Reduced = 282.52 kg

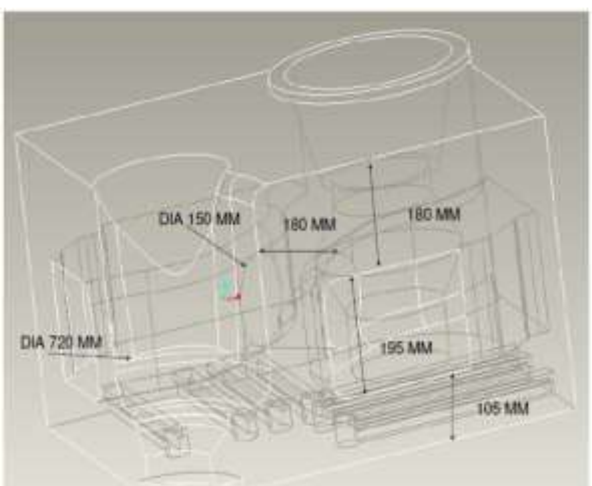

Fig 3.3.Iteration (3) Mass Reduced = 227.04 kg

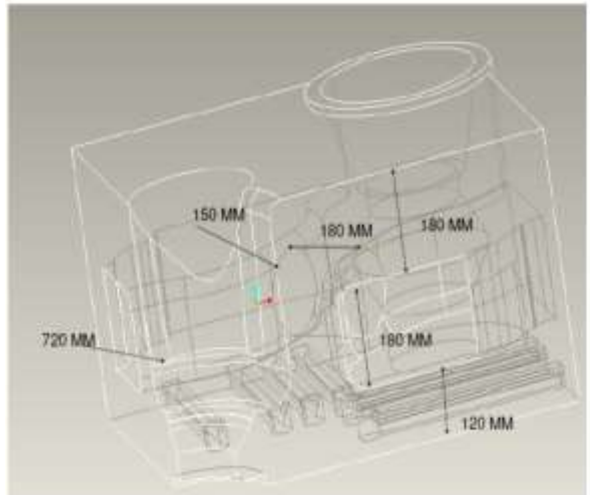

Fig 3.4.Iteration (4) Mass Reduced = 95 kg

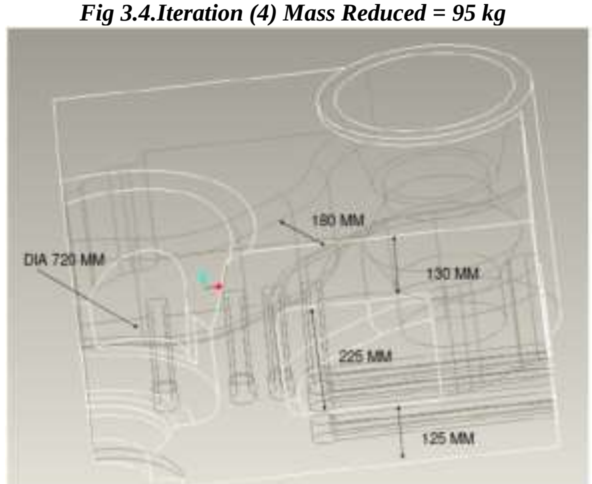

Fig 3.5.Iteration (5) Mass Reduced = 460.6 Kg

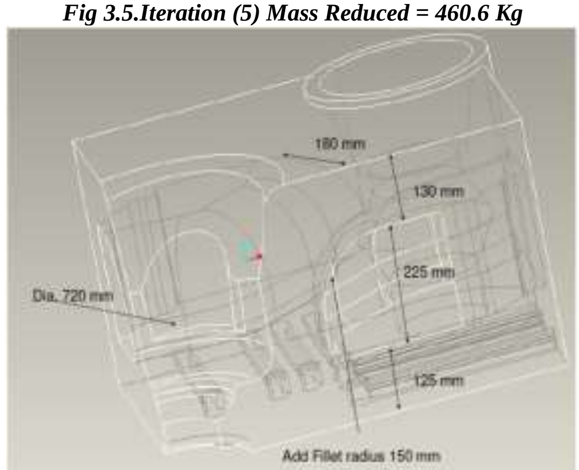

Fig 3.6.Final optimized Iteration (6) Mass reduced = 275.52 Kg

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

Justifications for each Iteration

- There are much other iteration carried out for design optimization but we are showing final six iteration which gives the best result.
- Maximum stress developed at then nozzle slot because nozzle slot acts as a stress raiser.
- The stress concentration factor is very high at nozzle slot.
- There are following iteration carried out for optimization of stationary platen
- In Iteration (1):- By providing fillet at Nozzle slot for reducing stress concentration.
- In Iteration (2):- By providing boss at upper side of Nozzle slot for minimizing stress concentration.
- In Iteration (3) & (4) :- By providing fillet and Revolve cut at Nozzle slot and try to convert Nozzle section in to circular or elliptical shape in this way stress concentration can be minimize.
- In Iteration (5) :- Further applying bigger radius revolve cut and fillet at Nozzle slot In this way try to reduce stress produce at Nozzle slot.
- Iteration (6):- Keep the revolve cut and fillet same as the iteration (5). But add more material added by applying fillet at pockets. Due to this the stresses and deflection values can be minimized at satisfactory level. It is considered final optimized stationary platen.
- Iteration (6) gives following stress, Deflection and weight reduction.
  - 1st principal stress (minimum mold case) = 99.073 N / mm2
    Deflection = 0.5254 mm
  - 1st principal stress (Vertical mold case) = 105.13 N / mm2
    Deflection = 0.4926 mm
  - 1stprincipal stress (horizontal mold case) = 68.66 N / mm2
    Deflection = 0.4570 mm
  - Weight reduction = 275.52 Kg.

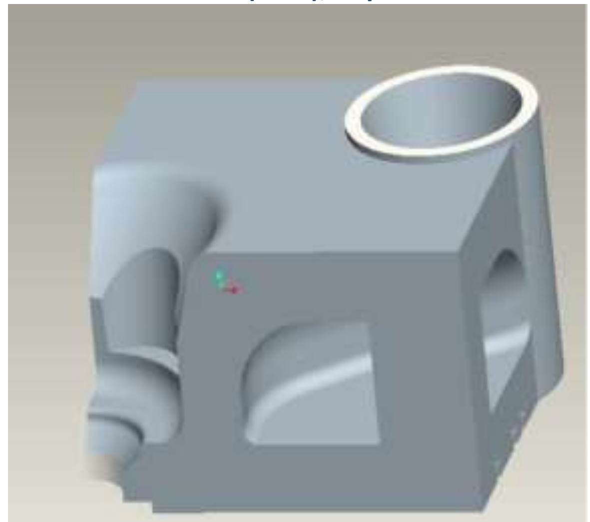

Fig 3.7.Optimized model

- Volume = 1.8893679e+08 mm^3
- Surface area = 3.9688364e+06 mm^2
- Density = 7.1000000e-09 tonne / mm^3
- Mass = 1.3414512e+00 tonne

Result

- 68.88 × 4 = 275.52 kg Mass reduced
- Percentage of weight reduction= 5 %

Analysis of Optimized Stationary Platen

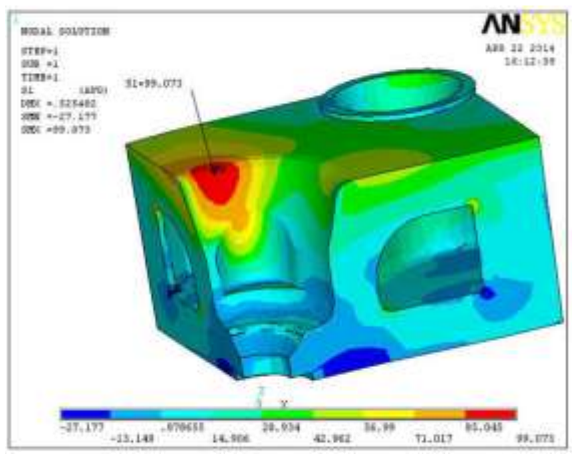

Fig 3.8.First principal Stress (Minimum Mold Case)

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

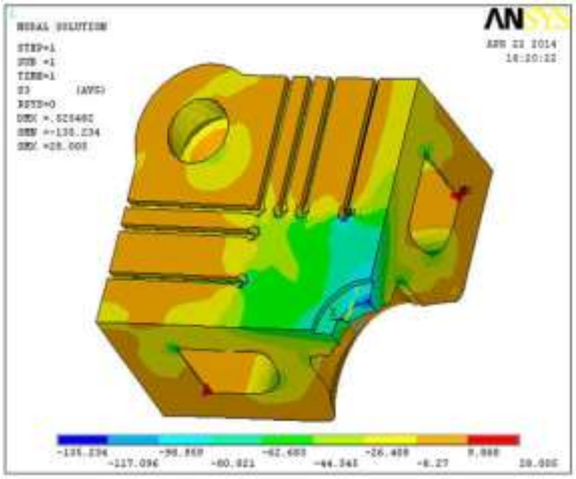

Fig 3.9.Third principal Stress (Minimum Mold Case)

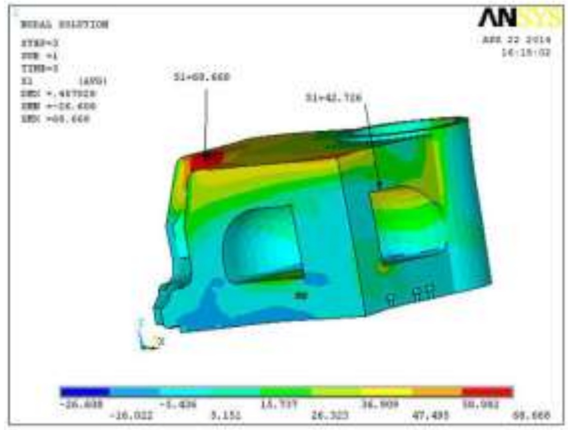

Fig 3.12.First principal Stress (Horizontal Mold Case)

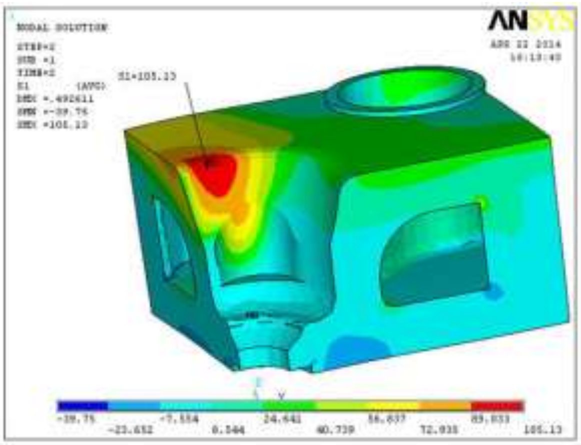

Fig 3.10.First principal Stress (Vertical Mold Case)

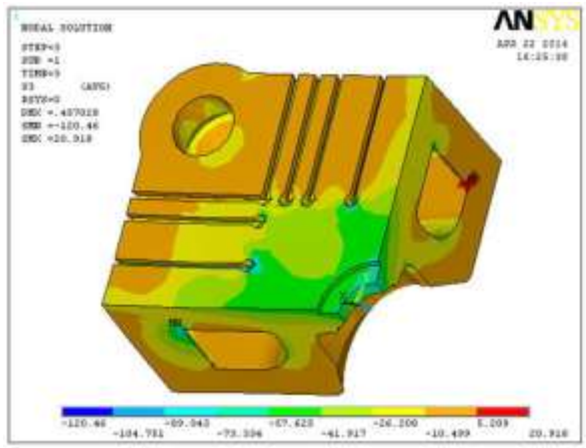

Fig 3.13.Third principal Stress (Horizontal Mold Case)

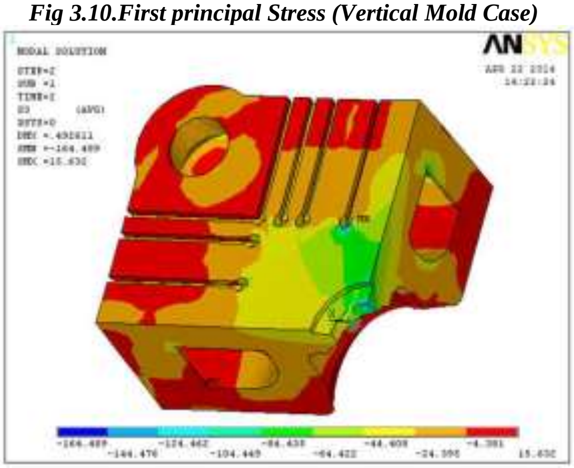

Fig 3.11.Third principal Stress (Vertical Mold Case)

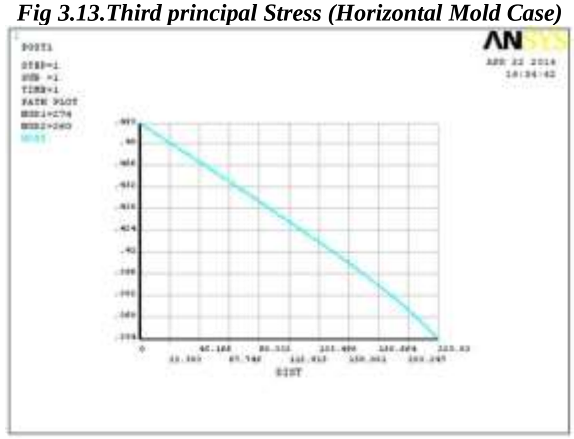

Fig 3.14.Horizontal relative Deflection for Minimum load case

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

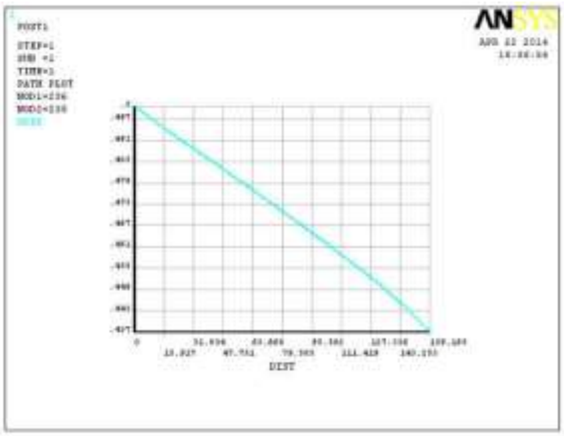

Fig 3.15.Vertical relative Deflection for Minimum load case

- Both horizontal and vertical relative deflections are within allowable limit.

Results Validation of Optimized Platen

<u>Final Optimization Results</u>

- 1st principal stress (minimum mold case) = 99.073 N / mm^2

  Deflection = 0.5254 mm.
- 1st principal stress (Vertical mold case) = 105.13 N / mm^2

  Deflection = 0.4926 mm.
- 1st principal stress (horizontal mold case)= 68.66 N / mm2

  Deflection= 0.4570 mm.
- Allowable stress: $\sigma_u \times K3 = S1$ N / mm^2 , Where K3 = constant.
- So above all induced stresses are less than the allowable stress
- All the stresses are coming under the design stress, and deflections are under the limit. Then results are correct.
- The Deflection Values are within limit.
- The relative deflection for horizontal and vertical are also within the limit.

Conclusion

- Casting and Machining Models of existing stationary platen has been created by using of Pro/E creo.
- FE analysis of existing stationary platen is carried out by using ANSYS software.
- T-slot and Nozzle slot have been included in the existing platen and FE analysis has performed for the existing stationary platen to incorporate with T slot and Nozzle slot.
- Maximum 1st principal stress is found to be at Nozzle slot but which is within the allowable limit.
- 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 %.
- Box type platen has other benefits also like it can be easily cast.

References

[1] *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.*

[2] *Dipl.-Ing. C. Hostert, Prof. Dr.-Ing. S. Maas, Prof. Dr.-Ing. A. Prof. Dr.-Ing. R.Nordmann, TU-Darmstadt; "Dynamic Simulation of an Injection Molding Machine"*

[3] *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.*

[4] *Yuvin Chinniah, Joseph-Jean Paques and Mathieu Champoux "Risk assessment and reduction A machine safety case study" Professional safety. Pages : 49-56, Oct 2007.*

[5] *S.Selvaraj,P.Venkataramaiah "Design and Fabrication of an Injection Moulding Tool" International Conference on Design and Manufacturing, Volume 64, Pages 1310-1319, 2013.*

[6] *Young. C. Waren, Roark's,"Formulas for Stress and Strain", 6th Edition, TATA McGraw Hill International edition.*

[7] *Johannaber, F., "Injection Molding Machines: a user's guide". München: Hanser Verlag 1994.*

[8] *Chandrashekhara, K. "Theory of plates"*, *page 50-56 Universities press, 3rd edition 2000*

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

[9] *Vardan, T.K.and Bhaskar, K., "Analysis of plates (Theory and problem)".page 3-14 Narosa Publishing house (1999)*

[10] *Steven M Lepi, "Practical guide to finite element: a solid mechanics approach" Publisher: New York, Marcel Dekker, ©1998.*

[11] *Cook, R. D., "Finite Element Modeling for Stress Analysis", J. Wiley & Sons, New York, 1995,*

[12] *Adams, V. and Askenazi, A., "Building Better Products with Finite Element Analysis on Word Press", Santa Fe, 1998. Moaveni, S., "Finite Element Analysis: Theory and Application With ANSYS" Prentice-Hall, Upper Saddle River, NJ, 1999, 2003*