

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 12, December 2018

Finite Element Analysis and Topology Optimization Design for Front Platen of Injection Molding Machine

Maulesh H Parikh¹, Krushal U Salaskar², Parth A Giri³, Aliasgar T Marchawala⁴, Bhavin R Patel⁵, Krunal B Patel⁶, Farhan S Ganam⁷, Meet Prajapati⁸, Meet Patel⁹, Yuvraj Raulji¹⁰

UG Scholar, Department of Mechanical Engineering, Gujarat Technological University,
Ahmedabad, India^{1, 2, 3, 4, 5, 6, 7, 8}

UG Scholar, Department of Mechanical Engineering, Parul University, Vadodara, India⁹

UG Scholar, Department of Mechanical Engineering, Navrachna University, Vadodara, India¹⁰

ABSTRACT: The weight of platens accounts for about 70% of the entire clamping device of injection molding machine. The reduction in the weight of platen means the reduction in the cost of injection molding machine. UG NX8.0 was adopted to conduct the parametric design, and UG NX8.0 “Advanced Simulation” module was adopted to conduct the finite element analysis and the topology optimization design for the front platen of injection molding machine so as to obtain the quantitative conclusions that the volume of platen is reduced and the stiffness as well as the strength is improved. The maximum deformation of platen is decreased by 6.2%, the volume is decreased by 6%, the maximum stress is decreased by 6.6% and the average displacement of all the nodes in the mold mounting plane is decreased by 9.1%. The effect of optimization is obvious.

KEYWORDS: injection molding machine; NX; platen; topology optimization; finite element analysis

I. INTRODUCTION

The platen is the main body of the mechanical structure in the injection molding machine. According to the information of each injection molding machine manufacturer, the weight of platens accounts for about 70% of the entire clamping device of injection molding machine. The reduction in the weight of platen means the reduction in the cost of injection molding machine. The structure and the load-carrying capability of the front platen of injection molding machine are more complex. The error of the approximation algorithm employed in engineering is larger so that the platen may become invalid in the process of using. In today’s rapid development of computer technology, there are more methods to change the current design situation. The finite element analysis can accurately obtain the stress distribution in the mechanical parts so as to improve the accuracy of calculation and the quality of design; the “checking design” is changed to initiatively find the better one and explore the optimal one through the topology optimization, which provides the theoretical basis for the improvement of the load-carrying capability and the structure design of platen. At the same time, it also provides an ideal tool for finding a platen structure with better stiffness and higher utilization rate of material.

II. TOPOLOGY OPTIMIZATION

The topology optimization is a shape optimization for the structure, which can be used to find the optimal material distribution scheme of bearing model under loading. For the topology optimization through using the finite element

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 12, December 2018

method, a pseudo-density η_i is assigned to each unit and η_i ($0 \leq \eta_i \leq 1$) is considered as the design variable to conduct multiple iterative calculation. When the objective function is optimized, the pseudo density of each unit will determine the fate of materials

NX8.0 is a powerful 3D digitizing software that combines with CAD/CAE/CAM. Its function covers the full life cycle of the product. The senior simulation module provides the topology optimization command. NX topology optimization employs the Tosca topology optimization solver produced in FE DESIGN Company. The design constraints (such as volume) as well as the manufacturing constraints can be defined.

The material distribution of the optimized model is massive. Afterwards, the model can become smooth through establishing a transitional model so that the distribution of the material can be more continuous. Then, the smoothed model (STL file) can be imported into the NX modeling as the reference of optimization design.

In this paper, the front platen of injection molding machine with 150 tons of clamping force has been conducted the finite element analysis and the topology optimization design through taking NX8.0 as the platform. The processes such as the platen design, the finite element analysis and the topology optimization are all performed in NX8.0.

III. FINITE ELEMENT ANALYSIS OF FRONT PLATEN

Establishment of the solid modeling of front platen. Considering that the institutions of the front platen of injection molding machine are not completely symmetrical, the whole model of the front platen has been established. Some holes and rounded corners are omitted in order to benefit the division of finite element mesh. The volume of this model is $0.06075m^3$.

Establishment of finite element model. As the structure of platen is complex, the 10-node 3D tetrahedral mesh is employed for division. The number of nodes of the whole model is 201889 and the number of units is 126400. The finite element model of the front platen before optimization is shown in Figure 1. The material of platen is QT500-7 ductile iron, the tensile strength is 500MPa, the yield strength is 320MPa, the elastic modulus is $E = 1.69 \times 10^5$ MPa and the Poisson's ratio is 0.275.

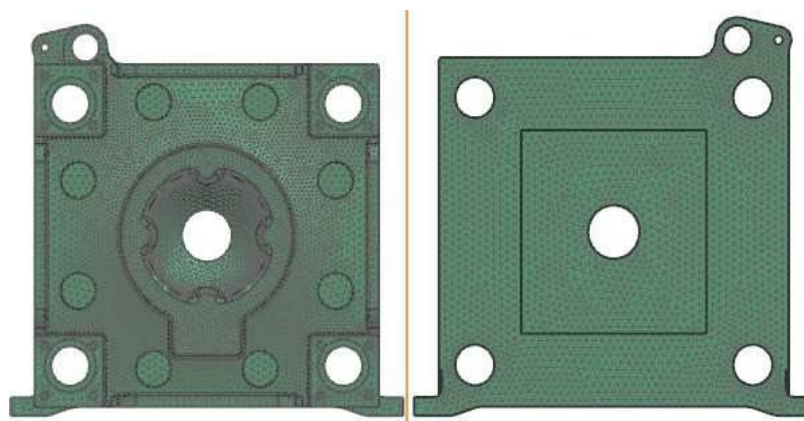


Figure 1 Finite element model of front platen before optimization

Constraints and loading settings. The four pull-rod end surfaces and the threaded holes of the front platen use the fixed constraint. The vertical displacement constraint is added into the two bottom rectangular planes. At the same time, 150 tons of clamping force are uniformly distributed in the rectangular mold mounting plane at the center of the back of platen, which is shown on the right of Figure 1.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 12, December 2018

Analysis of results. According to the result nephogram, it is found that the center hole in the back of platen has the maximum deformation, which is up to 0.130mm. The deformation is gradually decreased from center to outward. The average displacement of the 10288 nodes in the mold mounting plane is 0.055mm, which is shown in Figure 2. The maximum stress is 238.59MPa, which is appeared on the edge of pull-rod end surface.

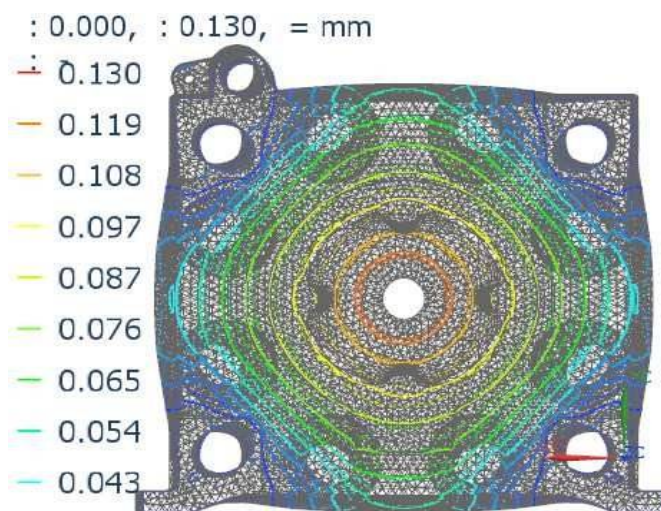


Figure 2 Contour map of platen deformation

IV. TOPOLOGY OPTIMIZATION DESIGN OF PLATEN

Model of optimization design. The purpose of the topology optimization is to find the optimal material distribution. Therefore, the prototype of platen optimization should be the solid with no ribs. The volume of this model is 0.06867. In order to ensure the shape of platen, the outline, the mold mounting plane, the center holes, the pull-rod holes and the contour of the end surface of pull-rod holes of platen should be not changed, which is shown in Figure 3. The loading constraints and the geometric constraints are not changed.

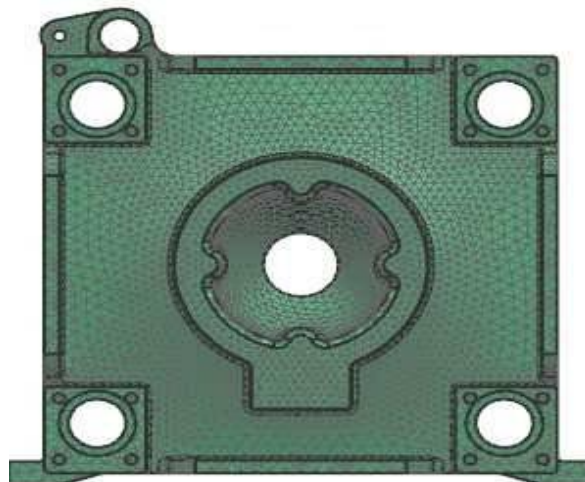


Figure 3 Topology optimization model of platen

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 12, December 2018

Analysis of topology optimization. For the topology optimization command of the senior simulation module of NX8.0, the settings are:

1. Defining all the units as the design area of topology optimization;
2. Defining processing and geometric constraints: the frozen frame, the mold mounting plane, the center hole, the pull-rod hole and the contour of the end surface of pull-rod hole;
3. Defining the design purpose: maximizing the stiffness;
4. Defining the design constraints: the volume does not exceed 80% of the solid platen with no ribs; the maximum displacement is no more than 90% of the one before optimization, namely 0.117mm.
5. It is required to set the smoothing parameters so as to ensure the smoothness of platen after optimization; the parameters should be in default; the recommended STL file can be generated after optimization and it can be imported to the NX model;
6. Defining the control parameters of topology optimization solver, such as the maximum iterations.

The results of solver are shown in Figure 4. The result map should be cut away 50% for convenient viewing. The unit with 1 density is shown as follows:

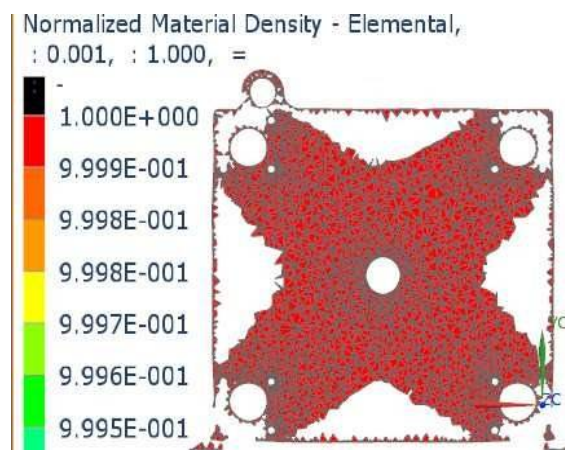


Figure 4 Material unit density map

Topology optimization design. In NX modeling, it is required to import the STL file generated in the topology optimization into the model. STL file is shown in File 5. The result map should be cut away 50% for convenient viewing.



Figure 5 STL model

The rib structure of platen should be re-optimized through taking the STL model as the reference, which is shown in Figure 6. The volume of model after optimization is 0.05713.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 12, December 2018

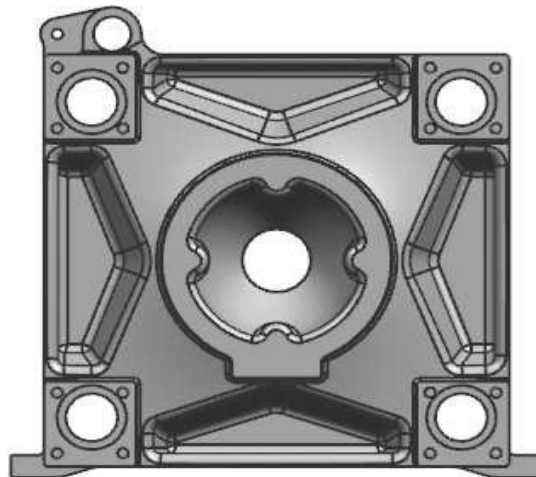


Figure 6 Optimized platen

Verification of finite element analysis. It is necessary to re-conduct the finite element analysis under the condition that the loading and constraint factors are not changed. The results are shown in Figure 7 and the maximum deformation is 0.122mm. The average displacement of the 10288 nodes in the mold mounting plane is 0.050mm. The maximum stress is 222.8MPa, which is appeared on the edge of pull-rod end surface.

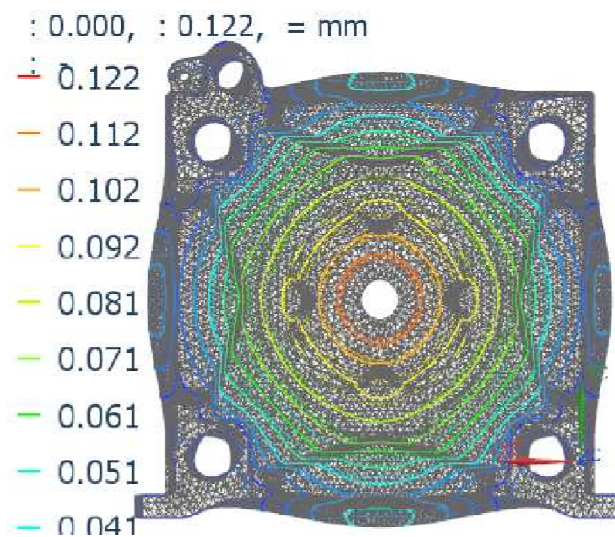


Figure 7 Contour map of platen deformation

Comparison of results

The comparison between the front platen after the topology optimization and the one before the optimization is shown in Table 1.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 12, December 2018

Table 1. Comparison of the results before and after optimization

	Volume (m^3)	Maximum deformation (mm)	Maximum stress (Mpa)	Average displacement of all the nodes in mold mounting plane(mm)
Before optimization	0.06075	0.130	238.59	0.055
After optimization	0.05713	0.122	222.8	0.050
Proportion of change	Decreasing 6%	Decreasing 6.2%	Decreasing 6.6%	Decreasing 9.1%

In Table 1, it is found that the maximum deformation of platen is decreased by 6.2%, the volume is decreased by 6%, the maximum stress is decreased by 6.6% and the average displacement of all the nodes in the mold mounting plane is decreased by 9.1%. The effect of optimization is obvious.

V. CONCLUSION

According to the comparison of results, it is believed that the optimization goals that the volume is decreased, the stiffness is increased and the strength is enhanced can be realized through the topology optimization design. As the modeling, the finite element analysis and the optimization design are performed in the same software, the seamless connection of CAD/CAE can be achieved, which ensures the unity of data and improves the edit ability of model.

REFERENCES

- [1] Rao S.S, "Engineering optimization ", Theory and Practice. Pearson Education.2003.
- [2] Bendsoe, M. P. and Sigmund O, "Topology Optimization - Theory, Methods and Applications", Springer, 2004.
- [3] N. Bakhtiary, "A new approach for sizing, shape and topology optimization", International congress and exposition, 1996.
- [4] B. Torstenfelt, A. Klarbring, Conceptual optimal design for modular car product families using simultaneous size, shape and topology optimization", Finite elements in analysis and design, vol. 43, pp.1050-1061, June 2007.