IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 3, Issue 1, Feb-Mar, 2015
ISSN: 2320 – 8791 (Impact Factor: 1.479)
www.ijreat.org

Topology Optimization Of Fixture Used For Quill Mounting

[1]Jasmin Bhimani, [2]Sanket Pandya, [3]Tejas Bhatt, [4]Ankur Vachhani

[1,2]Mechanical Engineering Department, V.V.P. Engineering College,

[3,4]Mechanical Engineering Department, B.H. Gardi College of Engineering, Rajkot, Gujarat-360005, India

Abstract

To achieve quick product development and cost effective product, manufacturing industries are continuously striving for various design and optimization tools like Topology Optimization, Size optimization, Shape Optimization, etc. In this paper focus is to minimize the weight of Fixture which is used to guide as well as locate the Quill. To analyze the fixture, Altair Hyperworks software is used. For Topology optimization OptiStruct tool is used to optimize the fixture within given constrains. Two concepts are used to optimize the fixture, i.e. Topology optimization with Draw direction concept and Without draw direction concept. The result shows that the new design having 50% reductions in weight compare to old one.

Keywords: *Fixture, topology optimization, quill, OptiStruct, Hyper works, static analysis*

1 Introduction

In this investigation focus was on Fixture which was designed to guide and locate the Quill. The quill is used to mount the Grinding wheel. Based on the applications and process parameters there are mainly two different sized Quill. The main objective is to reduce the weight of fixture to reduce operator fatigue.

In this paper, Static Analysis Topology Optimization is performed without compromising design functionality and reliability of Fixture. To perform Topology Optimization, OptiStruct tool available in Hyperworks software is used.

1.1 Static Analysis of Fixture

Static analysis was performed to find out extreme condition of Stress and deformation, while load due to self-weight of Quill was acting on Fixture. CAD geometry was prepared in CREO 2.0 and the model was imported in HyperWorks to carry out static analysis. The meshing was carried out using HyperMesh. Boundary conditions and loading conditions are applied and Finite Element Analysis was performed.

1.2 Topology Optimization

The manufacturing enterprises are continuously striving to develop the products cost effectively, having less weight and ensuring that products meet design functionality and reliability. To develop product faster, various optimization technique like Topology Optimization, Topography Optimization, Shape Optimization, Shape Optimization, Size Optimization are becoming more attractive.

The purpose of study is to minimize the weight of Fixture which was made using Nylon 6.To perform Topology Optimization, the prime requirement is Finite Element Analysis which provide the necessary data like maximum deformation, stress concentration, etc.

Topology Optimization without draw direction and Topology Optimization with draw direction concepts are used to obtain better result as well as manufacturable shape of Fixture.

1.3 Material Selection

There are various materials available in market which satisfies the design requirements. The Quill is critical part and metal to metal contact should be eliminated. Nylon 6 fulfils the design requirements also it is cost effective. So, Nylon 6 is selected.

Mechanical properties of Nylon 6 are given below.

Table 1 Nylon 6 material property

Modulus of Elasticity	8.3×10^9 N/m^2
Shear Modulus	3.2×10^8 N/m^2
Poision's Ratio	0.28
Mass density	1400 kg/m^3
Yield Strength	13.9×10^8 N/m^2

2 Literature review

Jagadish Baridabad et. al. [1] discusses about the project carried out at the Infotech Enterprises Limited on Altair OptiStruct, which involves optimization, carried out on RAT assembly. RAT Assembly used in the all commercial and military aircrafts. Primarily it is an air turbine device on which an aircraft is deployed into the air stream to deliver emergency hydraulic and electrical power when the aircraft's engine driven pumps and generators are unavailable due to a lack of engine power. The process of topology optimization begins with developing the equivalent FE model in HyperWorks and by understanding the modal analysis of existing design of the RAT. A modal analysis was performed to find the natural frequency of the RAT Assembly. This analysis included finding the natural frequencies of the RAT and comparison with the parent CAE software. Since their aim was to optimize the weight of the RAT assembly, it was then decided to carry out the topology optimization on the strut.

The setup for topology optimization is listed below:

1. Defining designable and non-designable space in the model

2. Creating the responses required to define global objective and constraints.

3. Design objective - Minimization of the Volume by Volume Fraction of designed space is taken as 0.75

4. Design Constraints - Natural Frequencies,

They have reduced 25% volume of designed space of strut leg by analysis and obtaining density plot for strut leg. So, this paper illustrates how the topology optimization tools can be used in the structural design of aerospace components. By attempting topology optimization, optimum material layout for maximum stiffness with reduction of weight of the any part can be generated and results obtained are comparable with parent CAE results.

Anton Olason et. al. [2] refers to investigate how and when structural optimization should be applied in the design process. They have used HyperMesh, Optistruct and HyperView software tools for optimization. They had taken a task reduce mass with maintained mechanical properties for given limitations. It was used to develop a sensible methodology together with guidelines for practical matters such as parameter values and recommended options. They have used three different software tools to complete the optimization. These were Hypermesh to discretize the model, Optistruct to solve the problem and Hyperview to evaluate the results. To know how optimization can be used in the design process they have taken a couple of different trial cases and evaluated them. One was of topology optimization of fixture designed to hold a test object of weight 1 kg during vibration testing and second was clip used to hold a circuit board in place.

To understand the effect of parameters for topology optimization, a parameter study was conducted. The parameters studied were volume fraction, minimum member size, penalization of intermediate densities, stress constraint in topology optimization, robustness etc. They had worked out two stages of the design process that are essentially in process of structural optimization. To develop a robust and usable methodology for topology optimization they had considered experience and conclusions from the trial cases and parameter study together with thoughts of designers and engineers. The methodology they have developed is shown in **Fig. 1**

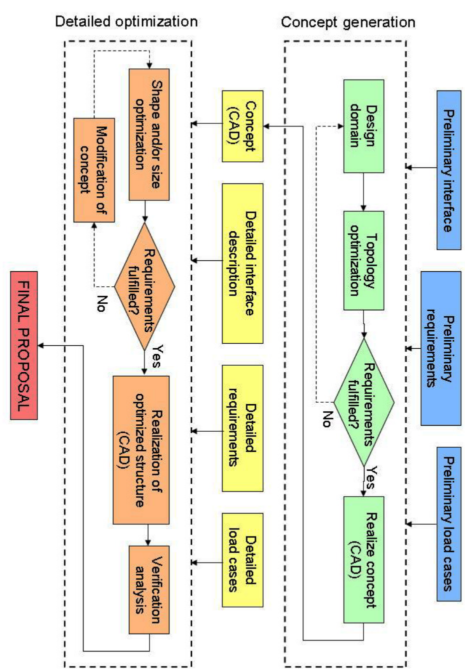

Figure 1 Methodology adopted for Optimization

Mohsen Ghaemi [3] has presented probabilistically constrained topology optimization problem in this thesis. The compliance approximation gives an appropriate tool to describe probabilistically given loading. The introduced algorithm provides an iterative tool which permits utilizing a large number of design variables what is unimaginable by the utilization of conventional stochastic optimization program. By the use of secondary meshing of ground elements the amount of the checker-board patter was stayed away from an acceptable level.

The present phase of the exploration demonstrates that to compute the stochastic topology needs the same computational time as if there is an occurrence of deterministic topology. The expounded topology design method is more effective when the relationship is not critical (smaller than 10% of the mean values). The algorithm is somewhat steady and provides the convergence to reach the optimum. One can see that the covariance values and the minimum probability values have huge impact on the optimal topology. The symmetry of the design can be lost because of the impact of the covariance values. The applied method gives a wider possibility to the designer to look into more reasonable loading description than the deterministic topology design.

Mahesh P. Sharma et. al. [4] have designed a knuckle which accommodates dual caliper mountings for increasing braking efficiency & reducing a stopping distance of a vehicle. Shape optimization method utilized as a part of this study in reducing the mass of knuckle by 19.35%. Additionally factory of safety is between 3 to 4. Maximum stress and displacement are within control. This optimization process also gives little change on the displacement. It implies that change of volume and shapes doesn't impact altogether to stiffness of the structure. Consequently, the overall weight of the vehicle can be decreased to attain to investment funds in expenses and materials, in addition, enhance fuel efficiency and lessen carbon outflows to maintain the nature.

Emil Norberg et. al. [5] has used topology optimization for beam and rod to achieve high stiffness of front structure of car. Author gives the idea on how to use Optistruct for topology optimization using various parameters. The effects of various parameters on topology optimization are given in this thesis.

3 Methodology adopted

3.1 Designing a CAD model of fixture and quill

CAD model of quill and fixture was develop in solid modelling software, Creo 2.0 considering ease of access. Fig. 2 shows CAD model of quill and Fig. 3 shows fixture of quill.

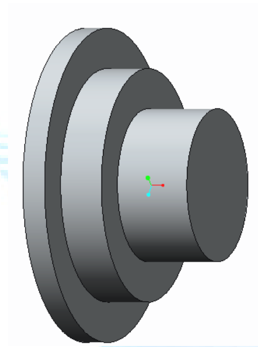

Figure 2 CAD model of quill

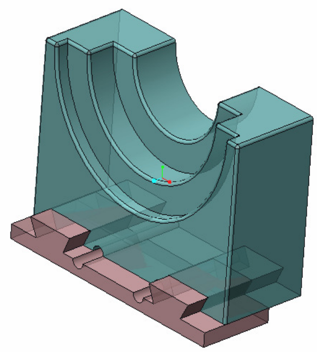

Figure 3 CAD model of fixture

3.2 Static analysis of fixture

To perform static analysis, CAD model is imported in Hyperworks. The meshing operation was performed on CAD model. The boundary condition and loading conditions were applied. Considering the maximum weight of quill, the force acting on surface of fixture was 1200N. There were 987 nodes on the surface, so 1.2N load per node were applied. Finite Element Analysis was performed.

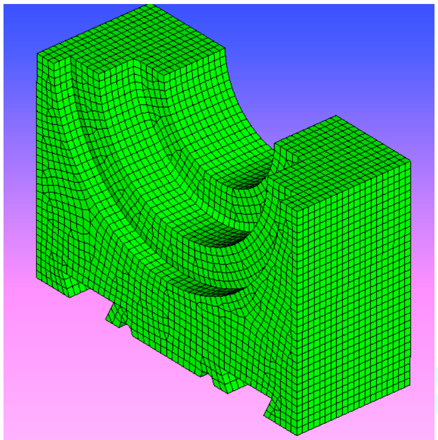

Figure 4 Mesh generation of fixture

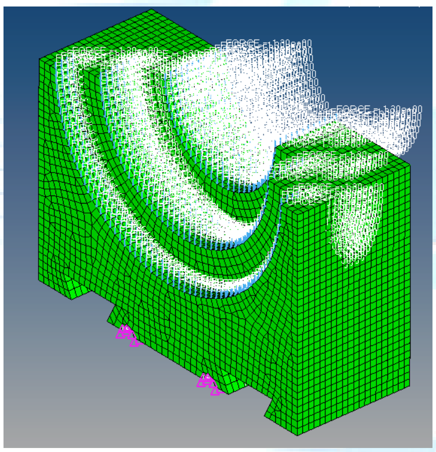

Figure 5 Boundary and loading condition

Fig. 4 shows mesh generation of fixture in Hypermesh software. There were total 24603 nodes and 22109 elements. Boundary condition and loading condition is depicted in Fig. 5. Pink color indicates boundary condition and white condition indicates loading condition.

3.3 Topology optimization

In this investigation, topology optimization was performed by using OptiStruct tool available in Hyperworks software. To achieve better result and manufacturable shape two different concepts were used, i.e. ***topology optimization without draw direction*** and ***topology optimization with draw direction.***

4 Result and discussion

The finite element model having von-mises stress 1.51 x 10^5 N/m^2 is shown in Fig. 6.

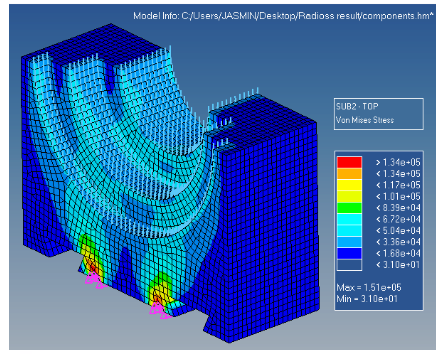

Figure 6 von-mises stress

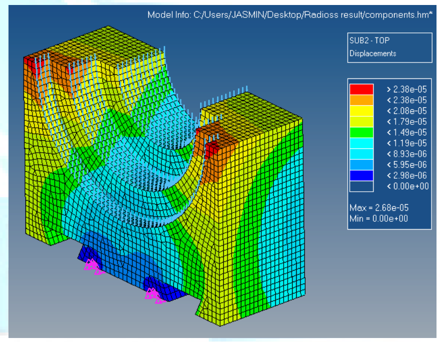

Figure 7 Maximum displacement model

Maximum displacement model is depicted in Fig. 7. The value of maximum displacement is 2.6x10^{-5} m.

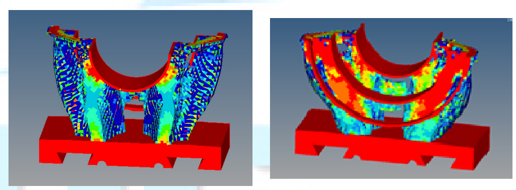

Figure 8 Optimized model without draw direction

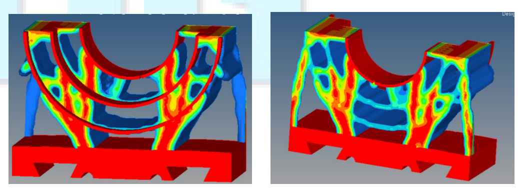

Figure 9 Optimized model with draw direction

	CAD model	Optimized model	% Reduction
Displacement	0.0268 mm	0.1 mm	
Von-mises Stress	0.151 N/mm^2	0.203 N/mm^2	
Mass	22 kg	12 kg	50%

Fig. 8 shows optimized model without draw direction concept and Fig. 9 shows optimized model with draw direction concept.

5 Summary

The Optimized model is having the weight of 12 kg when ***draw direction concept*** is used and 11 kg when ***without draw direction concept*** is used. The result obtained using with draw direction concept is easy to manufacture and give symmetric design. The design obtained after optimization were again analyse and it found to be safe. The % reduction in weight is approximately 50 % without compromising design functionality and reliability.

6 References

[1] Jagadish Baridabad, Vamsi Rajkumar and Ramakrishna Janamanchi, "Topology Optimization of RAT Assembly by the application of OptiStruct Tool", Altair Technology Conference-2013.

[2] Anton Olason and Daniel Tidman, "Methodology for Topology and Shape Optimization in the Design Process", Master's Thesis in the Master's programme Solid and Fluid Mechanics-2010:11, CHALMERS UNIVERSITY OF TECHNOLOGY, Göteborg, Sweden.

[3] Mohsen Ghaemi, "Topology optimization problems using optimality criteria methods", Budapestt Uniiverrsiitty off Technollogy and Economiics, 2009.

[4] Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi and Devendra A. Patel, "Static Analysis of Steering Knuckle and Its Shape Optimization", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 34-38.

[5] Emil Norberg and Sebastian Lovgren, "Topology Optimization of Vehicle Body Structure for improved Ride & Handling".

[6] Singiresu S. Rao. Engineering Optimization | Theory and Practice. John Wiley & ons, third edition, 1996.

[7] Altair Engineering Inc. Optistruct 10.0 help _les, 2009