International Journal of Engineering Sciences 2020 13(2) 58-64

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES

Topology Optimization of Frame of the Rack System

Rajesh Kumar Singh*, S. Ilavarasu#

*M Tech Student, Defence Institute of Advanced Technology (DU), Pune, India

#Scientist 'F', Centre for AirBorne System, DRDO, Bangalore, India

(Received 25 May 2020, accepted 28 July 2020)

Doi:https://doi.org/10.36224/ijes.130204

Abstract

Integration of the Electronics modules in the rack system has been very challenging and critical in nature. The functionality of the electronics module is of prime importance for the operation on any system or subsystem. In many cases where these electronics systems housed in a rack system which in turn mounted in aircraft fuselage make functioning of the system more severe. This paper deals with design optimization of frame of rack structure in line with increasing demand of more robust and light structure with features of high stiffness and structural integrity. Aerospace industries employ topology , size and shape optimization technique and have reported significant structural performance gains as a result ,This report deals with the topology optimization of frame of rack structure has been performed using OptiStruct software. The main objective is to find the optimal topology of frame of the rack. The rack structure has undergone the first level of optimization, in other word it is called finite element analysis with inertia loading condition which presents the stress contour with varying stress level, the stress contour highlight the maximum and minimum stress level in the structure which gives the first level of information about the material requirement within the structure. Topology optimization being the part of the structural optimization is the extended domain of the structural analysis where optimum placement of the material in the design space is the prime focus. The design space is a geometrical space where material alteration is effected to achieve the design objective. Topology optimization problems utilize the firmest mathematical basis, to account for improved weight-to-stiffness ratio and perceived aesthetic appeal of specific structural forms, enabling the Solid Isotropic Material with Penalization (SIMP) technique. Structural topology optimization is a technique for finding the optimum number, location and shape of opening with in the given design space of the rack structure to the series of loads and the boundary conditions. A range of topology of rack frame is obtained by setting varying the target volume fraction and an optimum topology of the frame is selected by satisfying stress to weight ratio requirement and manufacturing constraint.

Keywords: Topology optimization, Inertia loading, Compliance, Design domain, Solid Isotropic Material with Penalization

1. Introduction

There is an ever-increasing demand in the product development is to reduce weight, dimensions and stresses of structural profiles without degrading their basic properties such as rigidity, load-carrying capacity and general strength. The contemporary approach is to reduce material volume for individual structures while maintaining its basic mechanical and strength properties. A dominant trend which is more and more intensively developed is design optimization. The main objective of the optimization is to determine optimal and the most feasible design solution by presenting the best possible material layout within the design domain under applicable load and boundary condition. This approach is aimed at producing parts which are cheaper, more durable and have improved design characteristics. The optimization process provides design solution by considering constraints provide the design problem, however, with the use of the finite element method this process is relatively easy and requires minimum user's knowledge depending on how a given object is to be optimized. The Optimization process consists in determining a certain maximum or minimum of a function,

*Corresponding author

Email address: rajsinghmokama@gmail.com (Rajesh Kumar Singh)

ISSN 0976 – 6693.

while meeting specified conditions that limit a given problem. In topology optimization it is crucial that the region which is redundant in terms of excessive volume be effectively reduced by correctly defined loads and boundary conditions. There is a relatively great number of scientific works devoted to different optimization processes. The problem of optimization of topology, shape and parameters has been discussed in numerous research publications. The authors of the works [1, 2, 4,9] dealt with optimization of different criteria in mathematical terms, developing solutions based on numerical methods. The authors of the works [4, 3, 6, 8] solved the problems of optimizing physical processes using the finite element method.

2. Problem statement

The problem discussed in this paper pertains to a contemporary approach to structure analysis as topology optimization based on the use of the OptiStruct software. In the problem, there is frame member which is the part of a rack structure used for housing electronics modules. The frames of the rack structure are mounted on the base member whose flanges are riveted to mounting location. Therefore, the inertia load of the electronics module is transmitted through the frame member to the base of the rack structure as shown in fig. 1. Frame shown in *fig.* 2 being one of the main load transferring member, the basic task is to optimize the frame in order to have minimum compliance and weight under the restriction provide in terms of stress and displacement provided by static analysis of the frame carried out in the ANSYS software as shown in fig. 3

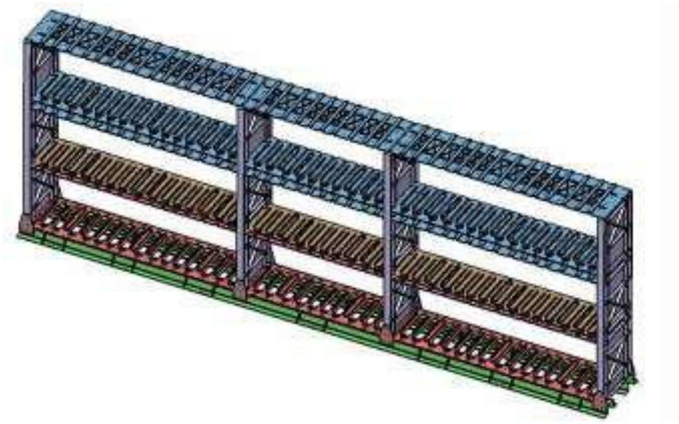

Figure 1: Rack structure

The process of design topology optimization is an innovative engineering approach which results in decreasing the structure's initial volume in the regions which are least stressed under the given load and constraint. The problem of profile optimization using the Optistruct system still requires a great deal of work and experience.

Figure 2: Frame CAD Model

Figure 3: Frame FE model

3. Topology optimization formulation

Topology optimization deals with converting the given design domain into design and non-design domain and it is desired to have an optimal placement of material in the design domain which is partitioned into void and solid elements by a finite element discretization.

The formulation is done by defining an 'objecting function', 'design variable', applicable 'design constraints' and design responses. For topology optimization, the objective function (compliance, volume fraction etc.) is either maximized or minimized subjected to design constraint where material density is acting as design variable

In mathematical terms, it is desired to have optimum placement of material which corresponds to (θ_{mat}) an optimal subset $\theta_{mat} \subset \theta$. Where θ is an available design domain. The design variable is represented by the density vector ρ containing elemental densities ρ_e. The local stiffness tensor E can be formulated by incorporating ρ as an integer formulation [7]

$$E(\rho) = \rho E^0 \tag{1}$$

$$\rho_e = \begin{cases} 1 \text{ if } e \in \theta_{mat} \\ 0 \text{ if } e \in \theta \backslash \theta_{mat} \end{cases}$$

And a volume constraint

$$\int \rho d\theta = \mathrm{Vol}\ (\theta_{mat}) \leq V \tag{2}$$

V is the total volume of the defined design domain. When $\rho_e = 1$, an element is considered to be a filled element whereas an element with $\rho_e = 0$ is considered to be a void element. This kind of formulation may lead to regular pattern of solid element and void which sometimes lead to as 'checkerboard problem'. To use a gradient based solution strategy for optimization problem, the integer problem described in (1) needs to be formulated as a continuous function so that the density function can take value between 0 and 1 [1]. The most common method to relax the integer problem is the SIMP (Solid Isotropic Material with Penalization) method.

The density function is then written as

$$E = \rho^p E^0,\ \rho \in [\rho_{min}, 1],\ p > 1 \tag{3}$$

Where 'p' is the 'penalizing factor' that penalizes elements with intermediate densities to approach 0 or 1, ρ_{min} is the lower density value limit to avoid singularities. Thus, the penalization is achieved without introducing any explicit penalization scheme. For materials with Poisson ratio $\nu = 0.3$, it is recommended in [2] to use $p \geq 3$.

The basic topology problem is set up in the following way:

The problem objective for the frame structure is to be as stiff as possible, while it is subjected to a certain reduced weight value.

Maximize Stiffness

$$\text{s.t. Vol}\ (\theta_{mat}) \leq V \tag{4}$$

Now assume linear elasticity, and replace stiffness by compliance to give a standard topology compliance optimization problem

Minimize Compliance, $C(\rho) = f^T u$ where u solves for the equilibrium equation $K(\rho)\,u = f$

$$\text{s.t.}\quad K(\rho) \quad u = f \tag{5}$$

Where, $K(\rho) = \sum_{e=1}^{n} \rho_e^p K_e^0$

and $K(\rho)$ is element stiffness matrix

Vol $(\theta_{mat}) \leq V$
$(\sigma)_{eqv} \leq \sigma_{ut}$*FOS

In this equation (5) the objective is to minimize the compliance. This objective can be achieved by varying specified design variable which is material density inside the design domain subjected to volume fraction and stress constraint

4. Finite element modeling

Finite Element model of frame is created in the HyperMesh software. Complete frame was meshed with 2D element (Shell 63). Finer mesh was created at the region near rivet holes and other critical region. Adequate care was taken for satisfying element quality parameter. Total 48488 quad and tria elements were used for the meshing the frame structure with minimum element size being 2 mm at the critical region. Finite Element model of the frame is shown in Fig. 4

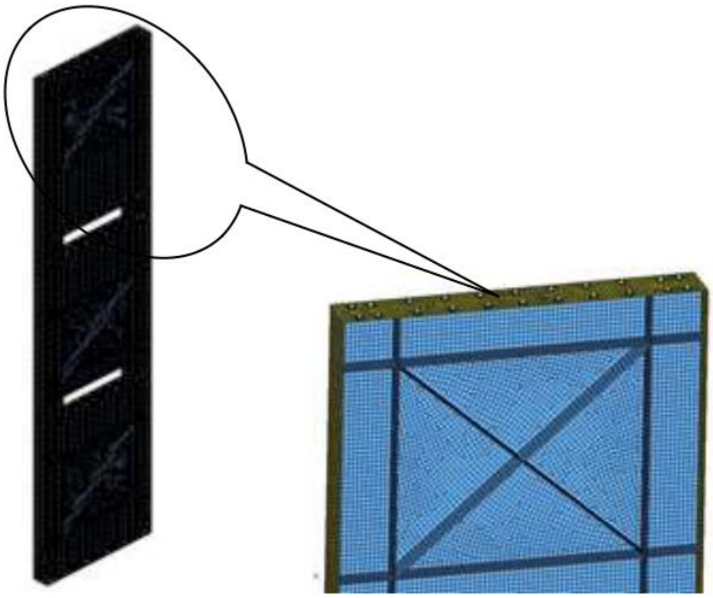

Figure 4: Frame FEA model

5. Load and boundary condition

The following figures show the load and moment acting at the fasteners location on various level of the frame resulting from the Finite Element analysis carried out using ANSYS software

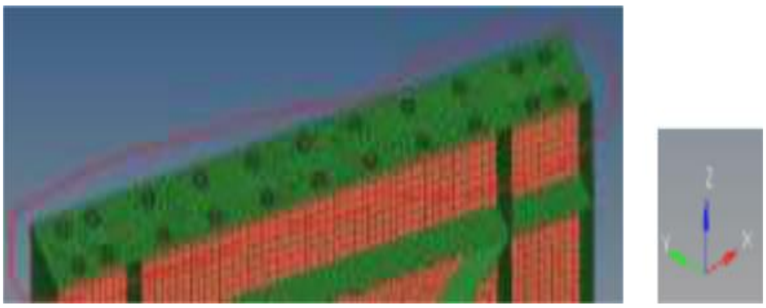

Figure 5: Loads & moment on frame top level

Force on each rivet center node F_Y = 3.3N, F_Z = 11N Moment on each rivet center node M_X = 652 Nmm

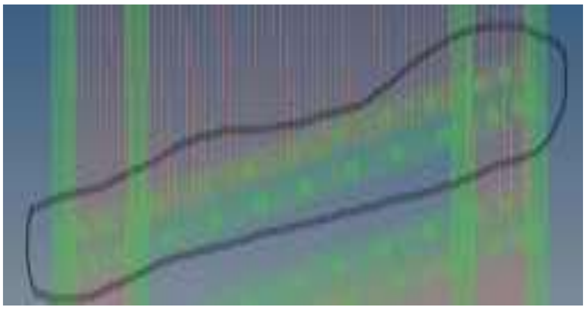

Figure 6: Loads & Moment on Frame third level

Force on each rivet center node F_Y = 10N, F_Z = 11N Moment on each rivet center node M_X = 410 Nmm

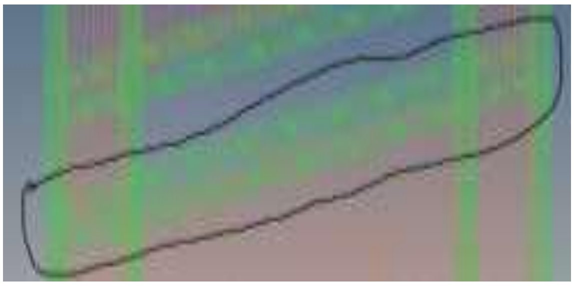

Figure 7: Loads & Moment on Frame second level

Force on each rivet center node F_Y = 10N, F_Z = 11N Moment on each rivet center node M_X = 367 Nmm

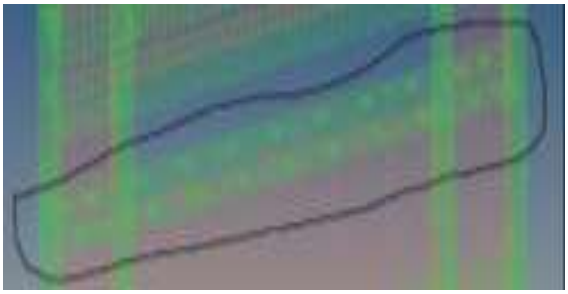

Figure 8: Loads & Moment on Frame first level

Force on each rivet center node F_Y = 2N, F_Z = 21N Moment on each rivet center node M_X = 136 Nmm

Bottom of the frame is riveted to the base structure which is idealized by restricting all Degree of freedom (DoF) at central nodes of the rivet locations.

6. Optimization plots

Topology optimization is carried in the OptiStrcut software as an objective to minimize compliance in order to have high stiffness subjected to constraint to 30% volume fraction and Von Mises stress should be less than 296Mpa. The following figure 9 shows the topology of the frame arrived at different iteration after optimally placement of masses within the design volume.

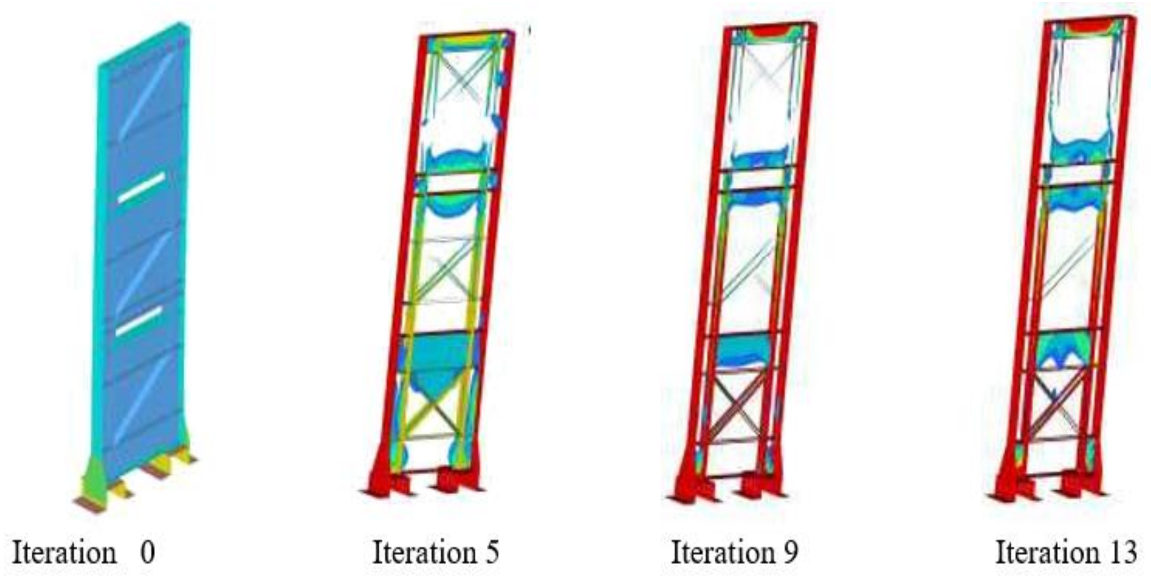

Figure 9: Topology of frame at different Iteration

7. Result and discussion

Referring figure 9 of Section VI, it is found that the optimum and feasible design is converged at iteration no 15. The geometrical topology obtained is complicated and very difficult to manufacture by considering the technological constraint, time and cost. The optimal geometry of frame is imported in the CAD software and the initial frame structure is modified as per the geometry suggested by topology analysis. It is also to be noted that the frame design is also simplified in line with optimal geometry such that it fame can be fabricated without much difficulties.

Frame is made up of Aluminum Alloy 2024 T351 having density (2.7 gm/cc), Ultimate stress 443 MPa and Young's Modulus 70GPa.The existing weight of the frame was 5.84 kg. and weight of the same frame after effecting geometrical changes as suggested by topology optimization comes out to be approx. 4.4 kg resulting weight reduction of approx. 1.44 kg per frame. As there are 04 frames in the rack assembly, therefore net weight reduction in the rack assembly is 6 kg.

8. Conclusion

The following conclusions can be drawn after carrying out topology optimization:

1. Topology optimization by finite element method provides optimal solution which can be analyzed based on the preset condition.
2. The pre and post result after optimization provides the detailed insight about the design and load transfer path and the same time it also helps in determining the process quality.
3. Reduction in structure volume reduces the weight of the structure and the same time the reduction of structure volume may lead to a slight increase in stresses.
4. Topology optimization generates only an approximate geometry of the structure which requires detailed judgement in terms of load carrying capacity, stiffness and manufacturability of the structure.

Acknowledgment

The author is grateful to the Director, CABS (DRDO), Head of Department, Mechanical Engineering Department, DIAT, Pune, Sri D R Makwana, Sc'E', VRDE (DRDO), Sri Vikas Kangude, and Sri Harsal Pingale, DIAT to guide and support the work.

References

[1] M. P. Bendsoe and O. Sigmund. "*Topology optimization: theory, methods and applications*". English. 2.;2; Berlin: Springer-Verlag, 2003;2004

[2] M. P. Bendsøe and O. Sigmund. "*Material interpolation schemes in topology optimization*". English. Archive of Applied Mechanics 69.9 (1999), 635–654

[3] Deaton, J.D. & Grandhi, R.V. "*A survey of structural and multidisciplinary continuum topology optimization*": Post 2000. Struct. Multidiscip. Optim., 2014, 49(1), 1–38.

[4] Guo, X. & Cheng, G.D. "*Recent development in structural design and optimization*". Acta Mech. Sin., 2010, 26(6), 807–23.

[5] Allaire, g.; Jouve, F. & Maillot, h. "*Topology optimization for minimum stress design with the homogenization method*". Struct. Multidiscip. Optim., 2004, 28 (2–3), 87– 98.

[6] Rietz, A. "Sufficiency of a finite exponent in SIMP (power law) methods. Struct. Multidiscip. Optim., 2001, 219(2), 159–63.

[7] Robin Larsson, "*Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm*", Chalmers Univarsity of Technology, G¨oteborg, Sweden 2016,

[8] Patryk Rozylo, "*Optimization of I-section profile design by the finite element method*", Advances in Science and Technology Research Journal, Volume 10, No. 29, March 2016, pages 52–56

[9] Optistrucut Design Manual